DOCUMENT RESUME

ED 339 881 CE 059 828

AUTHOR Skowronski, Steven D.; Tatum, Kenneth
TITLE CNC Turning Center Advanced Operations. Computer Numerical Control Operator/Programmer. 444-332.
INSTITUTION Milwaukee Area Technical Coll., Wis.
SPONS AGENCY Wisconsin State Board of Vocational, Technical, and Adult Education, Madison.
PUB DATE 90
NOTE 232p.; For a related manual, see CE 059 827.
PUB TYPE Guides - Classroom Use - Instructional Materials (For Learner) (051)

EDRS PRICE MF01/PC10 Plus Postage.
DESCRIPTORS Behavioral Objectives; Competency Based Education; *Computer Assisted Manufacturing; Computer Science Education; Electronic Control; Instructional Materials; Learning Activities; *Machine Tool Operators; Machine Tools; Manufacturing Industry; *Numerical Control; Programers; *Programing; Technical Education; Technical Institutes; Two Year Colleges

ABSTRACT

This student guide provides materials for a course designed to introduce the student to the operations and functions of a two-axis computer numerical control (CNC) turning center. The course consists of seven units. Unit 1 presents course expectations and syllabus, covers safety precautions, and describes the CNC turning center components, CNC control panel, and cathode ray tube (CRT). Unit 2 teaches manual operation of the machine tool axes and spindle for the purpose of homing or orientating the machine tool, manual tool loading and turret indexing, tail stock positioning, and soft key and CRT menu functions. Unit 3 prepares the operator to execute short programs created through the use of the Manual Data Input mode of the control panel and the exit mode. Unit 4 introduces a method of setting up the CNC turning center machine tool and program verification or prove-out procedures. Unit 5 introduces a new and unique concept in lathe tooling systems--Quick Setter and Quick Change Tool. Unit 6 instructs the student in canned cycle machining. Unit 7 introduces the set-up and operation of part programs that require the use of the tailstock for threading and grooving operations. Contents of each unit include an introduction, list of objectives, contents, and materials for three sections. These sections contain narrative instructional materials, an assignment section that requires written answers, a project section with instructions and necessary data sheets, and competency check list. (YLB)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

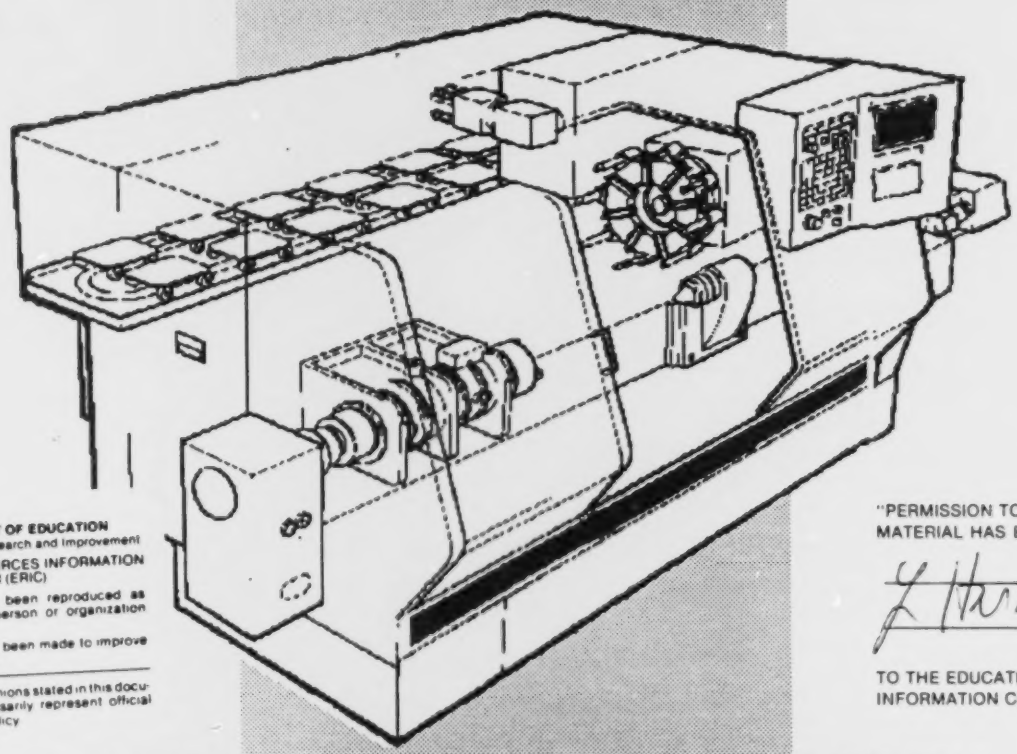
09-473

ED339881

matc Milwaukee Area Technical College

industrial division

CNC TURNING CENTER ADVANCED OPERATIONS



U.S. DEPARTMENT OF EDUCATION
Office of Educational Research and Improvement
EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)

✓ This document has been reproduced as received from the person or organization originating it.

☐ Minor changes have been made to improve reproduction quality.

• Points of view or opinions stated in this document do not necessarily represent official OERI position or policy.

"PERMISSION TO REPRODUCE THIS MATERIAL HAS BEEN GRANTED BY

L Hornbuck

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)."

COMPUTER NUMERICAL CONTROL OPERATOR/PROGRAMMER

444 - 332

By STEVEN D. SKOWRONSKI
and KENNETH TATUM

CE 059 828

BEST COPY AVAILABLE

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 1

Introduction To Machine Tool Components For The Two Axis Turning Center

INTRODUCTION

This course will introduce the student to the operations and functions of a two axis CNC turning center. This first unit will present the course expectations and syllabus. Next the description of the machine control panel will be covered with instruction on start up and shut down procedures. Procedures for orientating the machine axes and manually loading tools into the turret will follow.

The course will continue to progress through topics including machine lubrication and care, setup procedures, work piece chucking, program prove out, manual data input (MDI) and program editing.

OBJECTIVES

At the completion of this unit the student will be able to:

1. Discuss the course expectations and the course syllabus.
2. Locate and identify the various machine components of the turning center machine tool.
3. Power up the machine tool and power down according to manufacturers procedures.
4. Lubricate the machine tool according to scheduled maintenance recommendations.
5. Identify the various buttons , dials, and switches located on the control panel.
6. Discuss the safety precautions for machine tool operation.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

CONTENTS

Reference Section	Page No.
Section 1 Course syllabus	1 -
Section 2 Safety Precautions	1 -
Section 3 The CNC Turning Center Components	1 -
Section 4 CNC Control Panel And CRT	1 -

Assignment Section

On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section. 1 -

Project Section

Competency Evaluation 1 -

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

SECTION 1

COURSE SYLLABUS

Course Syllabus
CNC Turning Center Advanced Operations
(444-332)

Instructor:_____ Phone:_____

Office:_____ Office Hours:_____

Course Structure: 16 hours in CNC lab per week

Textbook: CNC Turning Center Advanced Operations
by Steven D. Skowronski (MATC Course 444-332)

Required Materials: Safety Glasses
12" Ruler

Reference Materials: HITEC-TURN 25S/25S-1100 CNC Lathe Instruction Manual

Grading Policy: Your final grade for this course is computed from the following:
25% assignments
25% competency evaluation
25% projects
25% quizzes

Attendance: Daily attendance in this course is mandatory. School policy for attendance will be enforced.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Week	Date	Unit of Instruction
1	____	1. Introduction To Machine Tool Components For The Two Axis Turning Center
2	____	2. Machine Control Unit Operation
3	____	3. MDI And The Edit Mode
4	____	4. Setup Methods
5	____	5. CNC Tooling Systems
6	____	6. Canned Cycle Machining
7	____	7. Threading And Tailstock Operation
8	____	Finish Projects And Final Exam

SECTION 2

SAFETY PRECAUTIONS

Safety precautions are important to protect the operator and the machine from accidents and damages respectively. These precautions apply not only to this machine but also to the settings and operations of almost all machines. Always observe these precautions.

General Precautions

Observing general precautions is greatly instrumental in creating an accident-free working environment and improving productivity.

1. Always wear safety eye glasses.

2. Do not operate the machine with gloves on.

3. Provide sufficient light around the machine and keep this area dry. Do not place obstacles around the machine and maintain a clean environment.

4. Do not use compressed air to clean and remove dust and cutting chips. from the machine proper, power control cabinet, NC unit and the floor around them.

5. Do not place a tool, workpiece and other things on any surface of the machine.

Precaution When Operating the Machine

Maintenance

1. The student/operator must fully observe the instructions on a caution plate. Take care not to stain, damage or remove the caution plate. When it cannot be read easily, contact the instructor.

2. Keep all doors and covers closed, except when adjusting. Exercise extreme care concerning the doors of the NC unit and the power control cabinet.

3. Do not remove or modify mechanisms and electric circuits installed for safety, such as limit switches for the stroke ends of travel axes, limit switches for the interlocking of a sliding cover, and so on.

4. For adjustment and repair, use the specified wrenches, and so on.

Operation

5. Remember the positions of the EMERGENCY STOP buttons so that you can push them immediately.

6. When starting the machine, follow the procedures mentioned later.

7. When the machine is in operation, never put your hand or head close to a rotating or moving section.

8. When removing stringy chips which wrapped around a tool or clog a chip conveyor or other areas, do not pull them with your hand. Use a brush or pliers to safely clear the chips. Be sure to stop the machine when disposing of the cutting chips.

9. Stop the machine when adjusting the position of a coolant nozzle.

Tool Setting

10. When setting up tools, stop the spindle as well as the feed in each axis.

11. Be very careful of tool noses when setting them up. Do not set the tools over their specified lengths because their tool noses may interfere with a bed, carriage, cover, tailstock, etc. when indexing a turret head.

12. Mount tools in a well balanced condition. Due to high-speed turret indexing (HITEC-TURN 25S: about 3 sec. for 5 faces), when their mounting is unbalanced, it may lead to improper turret indexing.

Workpiece Chucking

13. When chucking a workpiece, be careful of its balance. Do not run the spindle if the mounting of the workpiece is unbalanced.

14. Use standard soft jaws. Mount the jaws so that they may stay within the outer diameter of the chuck when possible. Otherwise check for adjacent tool interference.

15. For a chuck cylinder set pressure, determine it referring to PAGE() "Soft Jaw Forming and Chuck Gripping Force". Take note that the chuck gripping force will be suddenly reduced due to a centrifugal force when the spindle runs at a high speed.

16. When forming the soft jaws, pay full attention to a forming ring gripping position and a shape to which the jaws are to be formed. After forming, check that the jaws properly grip the workpiece and that the chucking pressure is adequate.

17. When chucking and centering a center workpiece, take special note of a workpiece weight, a center hole and a thrust force applied to the center hole. If a heavy workpiece is centered with a small center hole and a load is applied, the tip of the center may be damaged, allowing the

workpiece to jump out.

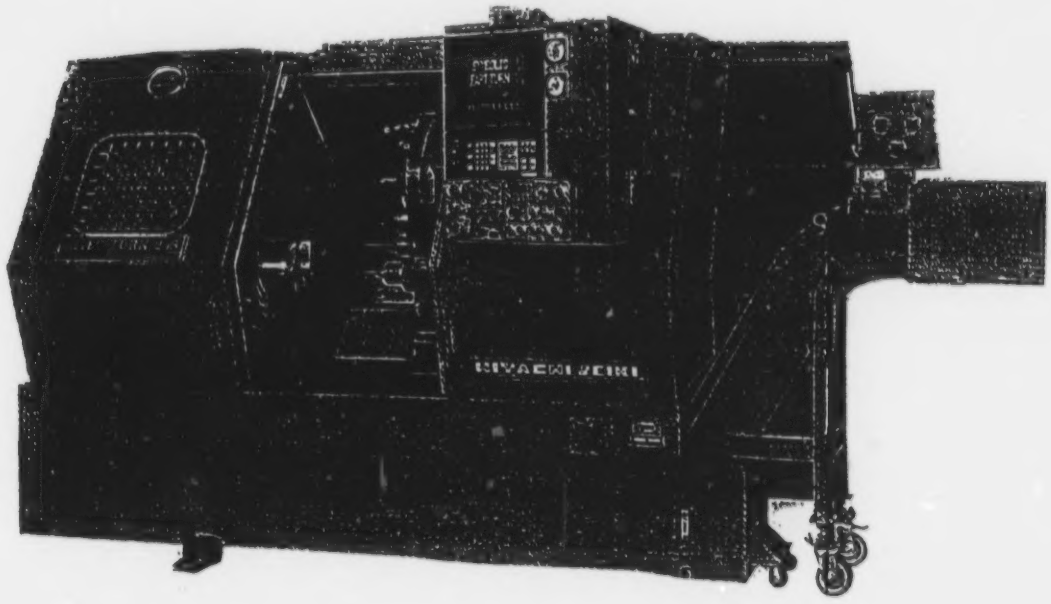
Completion of Daily Work

18. After finishing daily work, be sure to turn off the power, clear the machine and apply rust preventing oil on slide ways, etc. in accordance with the specified procedures. Carefully perform these operations when a water-soluble coolant is used.

SECTION 3

THE CNC TURNING CENTER COMPONENTS

Machine Outline



HITEC-TURN 25S SLANT BED CNC TURNING CENTER
Figure 1-1

This is an NC lathe with a drum type turret head. It is designed to be able to control two axes simultaneously. The following are the features of this machine.

1. The machine can perform chuck work and center work.

2. Spindle speeds can be steplessly changed in a wide range due to the employment of an AC invertor motor.

3. With a permanent tool setting as a basis, various kinds of workpieces can be machined without changing tool settings. Furthermore, tools available on the market can be used without modification.

4. As a standard feature, the machine has a constant surface speed control function.

5. The machine also has a diagnostic function.

In addition, the turret head is positioned in the rear of the machine in order to improve accessibility to the spindle and the operation of the machine.

Construction of Machine Proper

1. Head stock
   Spindle speeds are automatically changed in a wide range from 20 to 3,600 rpm by the AC motor and 2-step gear shift (hydraulic). The AC invertor motor makes it possible to perform constant surface speed control.

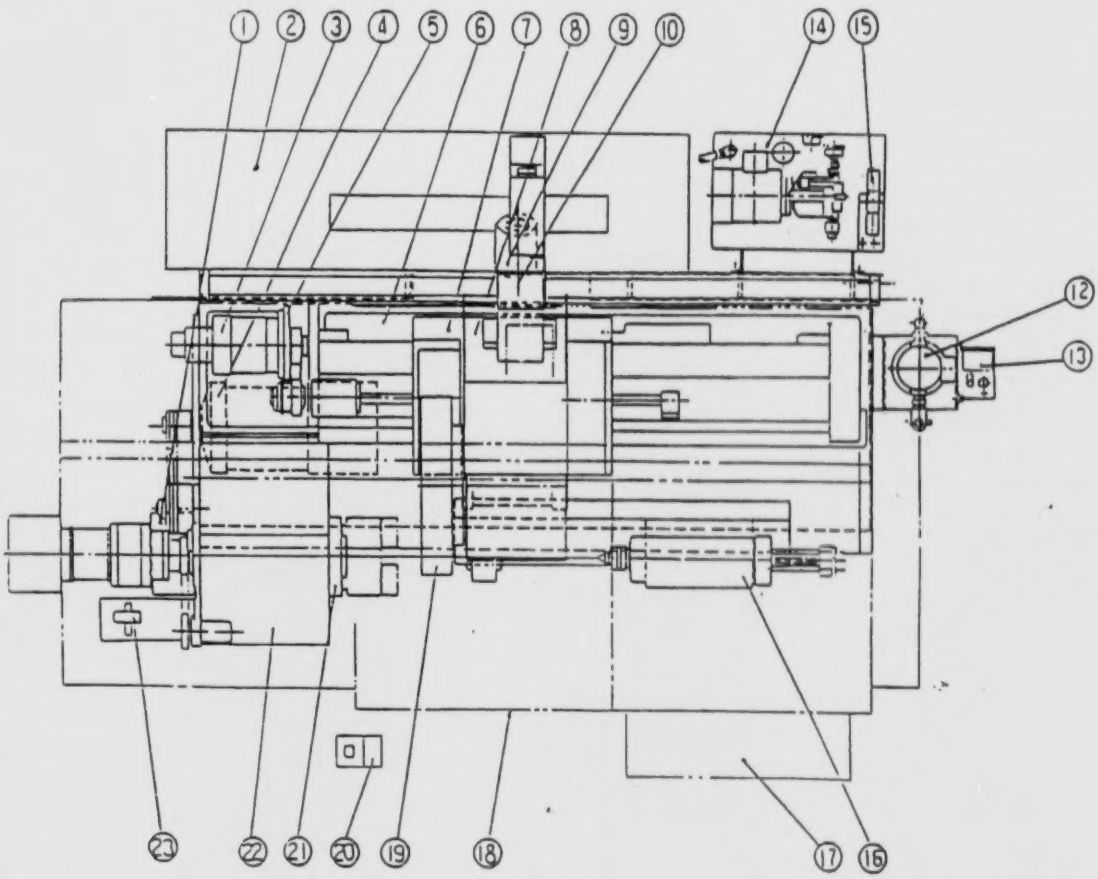
2. Turret head
   The turret head guarantees high indexing accuracy due to the introduction of a unique coupling system. The turret head is clamped and indexed hydraulically. In order to secure its performance, a rough indexing mechanism is incorporated.

3. Tailstock
   The taper of the tailstock is MT No. 5, and any of live centers on the market can be mounted. The tailstock can be set at an arbitrary position on a pedestal, and the tailstock spindle can be moved by

100 mm by pressing a button on the operation panel. A thrust force can be adjusted.

Machine Diagram

Compare the figure below with the actual machine tool and locate each section as listed on the next page.



TOP VIEW HITEC-TURN 25S SLANT BED CNC TURNING CENTER
Figure 1-2

Name of Each Section

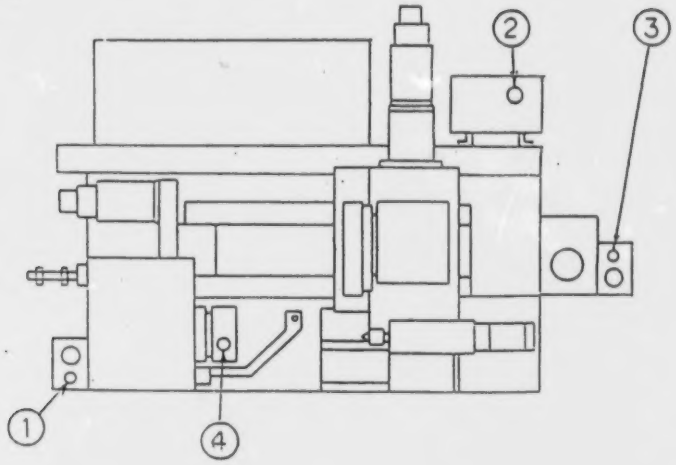
1. Headstock Lubricating Oil Tank
2. Power Control Cabinet/NC Unit
3. Z-axis Servo Motor
4. Spindle Motor
5. Z Feed Bracket
6. Bed
7. Saddle
8. Cross Slide
9. X-axis Servo Motor
10. X Feed Box
12. Coolant Pump
13. Lubricating Unit
14. Hydraulic Tank
15. Tailstock Pressure Control Manifold
16. Tailstock
17. Operation Panel
18. Sliding Cover
19. Turret Head
20. Chuck Open/Close Foot Switch
21. Spindle
22. Headstock
23. Chucking Pressure Control Manifold

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Specifications of Machine Proper		HITEC-TURN 25S (mm)	(in)
Chuck outer diameter		254	10
Max. machining length		600	23.6
Spindle	Diameter of spindle through hole	70.5	2.77
	Spindle nose	A2-8	
	Spindle speed (RPM)	20 --> 3600	
	No. of automatic speed change steps	Stepless	
Swing	Max. swing	510	20.0
Bed	Width upper surface	420	16.5
	Overall length	2160	85
Turret	Max. distance between spindle nose and OD tool nose	770	30.3
	Stroke in longitudinal direction (Z axis)	620	24.4
	Stroke in transverse direction (X axis)	235	9.3
	Rapid traverse in longitudinal direction (z axis)	12000 /min	480 /min
	Rapid traverse in transverse direction (X axis)	8000/min	315 /min

Specifications of Machine Proper		HITEC-TURN 25S (mm)	(in)
Turret	Feed rate: minimum	.001 /rev	.0001 /rev
	maximum	500.0	19.685
	No. of tool mounting faces	10 (for QCT tooling)	
	ID tool mounting hole pitch circle diameter	482	18.98
Tailstock	Diameter of tailstock spindle	90	3.54
	Taper of tail spindle	MT No. 5	
	Stroke of tailstock spindle	100	4
Motors	Spindle drive motor	AC 25/20 hp (30 minutes continuous)	
	Hydraulic unit drive	1 hp	
	Coolant pump drive	.53 hp	

Lubrication Chart



TOP VIEW OF THE HITEC-TURN 25S TURNING CENTER
Figure 1-3

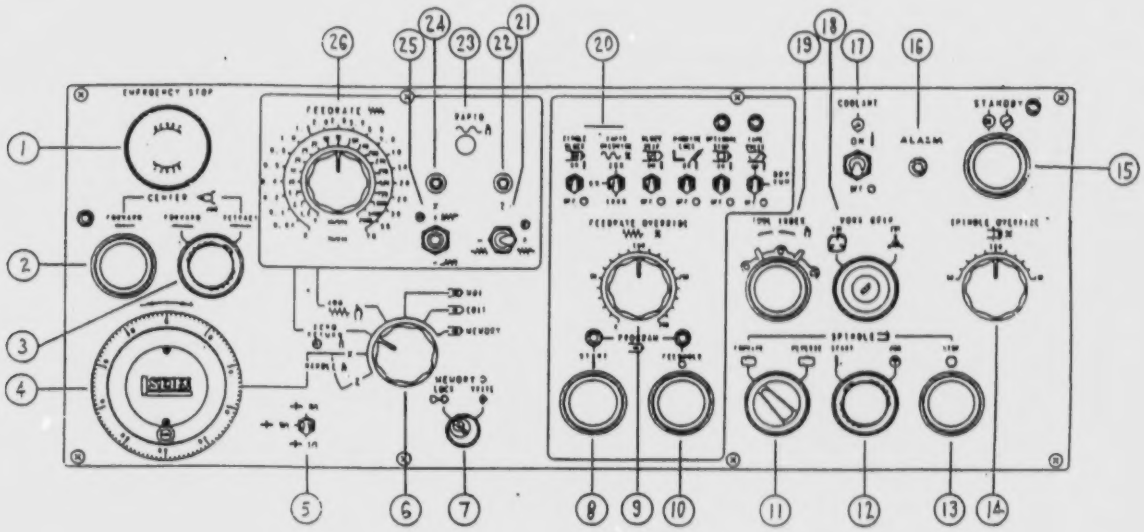
LUBRICATION POINT	METHOD	QUANTITY/FREQUENCY	OIL TYPE
1. Headstock	Trochoid pump	Fill to gage mark, filter or change oil every 6 months	Mobile DTE light oil
2. Hydraulic tank	Piston pump	Fill to gage mark, filter or change oil every 6 months	Mobile DTE light oil
3. Slideways	Forced lubrication	Check frequently or fill when LUBE ALARM lights up	Mobile Vactra oil No. 2
4. Hydraulic chuck	Manual lubrication	Fill with grease once a day	Dow Corning Molykote BR2-S grease

SECTION 4

CNC CONTROL PANEL AND CRT

Control Panel

Study the control panel below and compare it to the actual machine control panel. The function of each item is described below.



THE CNC CONTROL PANEL
Figure 1-4

1. EMERGENCY STOP BUTTON
   This button disables the machine operation.

2. CENTER FORWARD BUTTON AND LAMP
When the quill is stopped, pressed this button continuously moves the quill forward and turns on its indicator lamp.

3. CENTER JOG FORWARD/RETRACT SELECTOR BUTTON
The tailstock quill jogs forward or backward only while this button is pressed.

4. PULSE GENERATOR DIAL (HANDLE)
Handle feed is available when the mode select switch is shifted to the HANDLE X or Z position.

5. HANDLE FEED MAGNIFICATION SELECTOR SWITCH
This switch selects a feed rate per handle graduation; 1/1 for 0.001, 10/1 for 0.0101 and 100/1 for 0.100.

6. MODE SELECT SWITCH
HANDLE X, HANDLEZ, JOG, and ZERO RETURN are referred to as manual modes, and MDI, EDT, MEMORY, and TAPE as automatic modes.

7. MEMORY LOCK/RIGHT KEY
This key enables and disables the program editing operation.

8. PROGRAM START BUTTON AND LAMP
Pressing this button starts MDI or automatic operation.

9. FEED RATE OVERRIDE SWITCH
With this switch, a feed rate in the MDI mode can be changed in a range form 0% to 200%.

10. PROGRAM FEED HOLD BUTTON AND LAMP
Pressing this button suspends MDI or automatic operation. Only feed is decelerated and stopped.

11. SPINDLE FORWARD/REVERSE SELECTOR BUTTON

This button selects a spindle running direction in either a clockwise or counter-clockwise direction.

12. SPINDLE START/JOG SELECTOR BUTTON
START = The spindle runs when the mode select switch is set to a normal mode.

JOG = Regardless of the setting of the mode select switch, the spindle runs only while this button is pressed.

13. SPINDLE STOP BUTTON
When the mode select switch is shifted to the manual mode, pressing this switch will stop the spindle.

14. SPINDLE OVERRIDE SWITCH
This switch can override the programmed spindle speed from 50 to 150%.

15. STANDBY BUTTON AND LAMP
Operation enabling button and lamp.

16. ALARM LAMP
This lamp lights up in case of abnormality of the NC control unit, motor, or lubrication.

17. COOLANT ON/OFF SWITCH

18. WORK GRIP INT/EXT SELECTOR KEY
This key switch selects in which direction the chuck grips the workplace.

19. TOOL INDEX
When the mode select switch is shifted to the manual mode, pressing this button indexes the turret toolhead.

20. TOGGLE SWITCHES

SINGLE BLOCK
On = The machine stops its operation after executing one block of information. Pressing the START button proceeds to execute the next block of information.

Off = Continuous operation.

RAPID OVERRIDE
Select a safety speed 5%, 25%, or 100%.

BLOCK SKIP
On = A block containing "/"(slash) is skipped.

Off = A block containing "/"(slash) is executed.

MACHINE LOCK
On = The turret head stops at a present position and does not move. The rest of the machine functions.

Off = The turret head functions as commanded.

OPTIONAL STOP
On = M01 function is executed (lamp lights up). To restart, press the start button.

Off = Operation proceeds to the next block, skipping the M01.

DRY RUN
Tape Check = Spindle rotation is suspended in addition to the dry run function.

Dry Run = A manually set feed rate becomes effective, ignoring rapid traverse and cutting feed in the CNC program.

Off = The tool travels as commanded.

21. "Z" AXIS MANUAL CONTINUOUS FEED SWITCH.
Same function as the "X" axis manual continuous feed switch.

22 "Z" AXIS ZERO RETURN LAMP
This lamp lights up when the "Z" axis is located at the machine zero home position.

23. RAPID BUTTON
The machine operates at a percentage value set by the RAPID OVERRIDE switch only while this button is pressed during manual continuous feed.

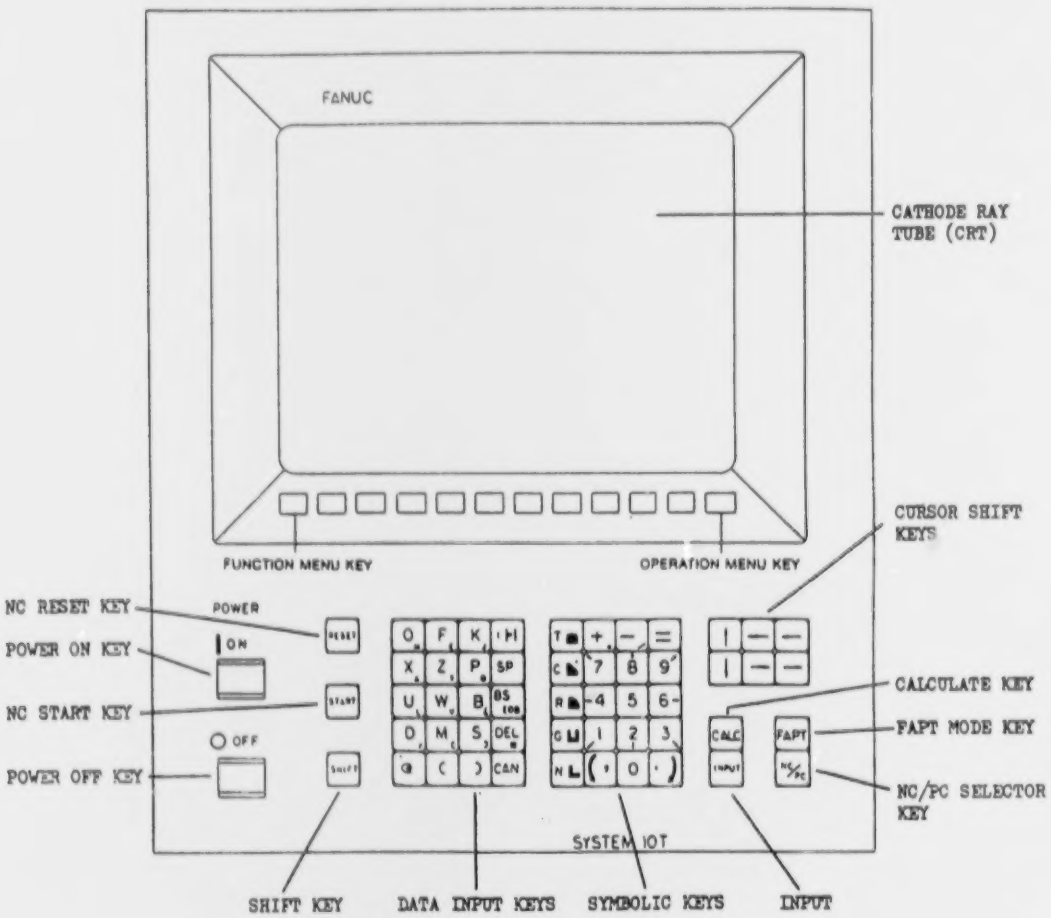
24. "X" AXIS ZERO RETURN LAMP
This lamp lights up when the "X" axis is located at the machine zero point.

25. "X" AXIS MANUAL CONTINUOUS FEED SWITCH
Manual continuous feed is available when the mode select switch is shifted to JOG.

26. FEED RATE SWITCH
This switch selects a feed rate for the manual continuous feed and dry run operations (tape check and dry run).

CRT Control Panel

Study the CRT control panel below and compare it to the actual machine CRT control panel. The function of each item is described below.



FANUC SYSTEM 10TE-F CONTROL PANEL
Figure 1-6

1. CATHODE-RAY TUBE (CRT)
   14" color screen.

2. POWER ON/OFF KEY
   This key turns the NC power on and off.

3. RESET KEY
   Press this key to reset the NC, to cancel an alarm, etc.

4. START KEY
   Press this key to start the MDI commands, or to start the automatic operation cycle.

5. SOFT KEYS
   The soft keys have various functions, according to the applications. The soft key functions are displayed at the bottom of the CRT screen.

6. FUNCTION MENU KEYS
   When the function menu key is pressed with the soft keys switched to function other than the function selection, the soft keys are brought back to the function selection. When the function menu key is pressed with the soft keys switched to the function selection, the soft keys are switched to function selection keys other than those currently shown on the CRT. The MDI and CRT panel is provided with 10 soft keys. However, these keys are not sufficient for some applications. In this case, sign + shown at the right end of the bottom line on the CRT indicates that there are some keys which have been indicated.

7. OPERATION MENU KEY
   The functions of the soft keys are changed in various ways according to applications. When the operation menu key is pressed with the soft keys switched to function other than the operation selection, the soft keys are switched to possible operation selection keys shown on the CRT. When the operation menu key is pressed with the soft keys switched to the operation selection, the soft keys are switched to the operation

selection keys other than those currently shown on the CRT. The MDI and CRT panel is provided with the 10 soft keys. However, these keys are not sufficient for some applications. In this case, the plus (+) sign shown at the right end of the bottom line on the CRT indicates that there are some keys which have not been indicated.

8. ADDRESS AND NUMERICAL KEYS
Press these keys to input alphabetic, numeric, and other characters.

9. SHIFT KEY
Some address keys have two characters on top. Press the shift key and an address key in that order. Then the character on the lower right can be input. When the shift key is pressed, ^ is displayed in the key input buffer. This indicates that the character on the lower right side of an address key can be input by pressing the address key.

10. INPUT KEY
When an address or a numerical key is pressed, data is input once to the key input buffer, and it is displayed on the CRT screen. To set the data input to the key input buffer in the offset register, etc., press the INPUT key. This key is equivalent to the INPUT keys of the soft keys, so the same result can obtained by pressing either of them.

11. CANCEL (CAN) KEY
Press this key to delete the last character or symbol input to the key input buffer. The contents of the key input buffer are displayed on the CRT screen. When the address key or the numerical key is pressed next, the position where the data is inserted next is indicated by "-". When the cancel (CAN) key is pressed, the character immediately before the "-" is cancelled.

EXAMPLE: When the key input buffer displays >N001X10.0Z_ and the cancel (CAN) key is pressed, Z is cancelled and >N001X10.0_ is displayed.

12. CURSOR SHIFT KEYS

There are four kinds of cursor shift keys described below.

-->: This key is used to shift the cursor a short distance in the forward direction.

<--: This key is used to shift the cursor a short distance in the reverse direction

-->>: This key is used to shift the cursor in the key input buffer, when the character(s) is input into it. The cursor position in the key input buffer is indicated by "-". "-" indicates the position where the data is inserted when the next address or numerical key is pressed. When the cancel (CAN) key is pressed, the character immediately before "-" is cancelled. When nothing is input to the key input buffer, this key is used to shift the cursor on the CRT screen a long distance in the forward direction.

<<--: When the character(s) is input to the key input buffer, this key is used to shift the cursor in the key input buffer. When nothing is input in the key input buffer, this key is used to shift the cursor on the CRT screen a long distance in the reverse direction.

13. PAGE CHANGEOVER KEY

Two kinds of page changeover keys are described below.

This key is used to change over the page on the CRT screen in the forward direction.

This key is used to change over the page on the CRT screen in the reverse direction.

14. CALCULATION (CALC) KEY

Press this key to execute the calculation instruction in the key input buffer.

EXAMPLE: When pressing the calculation key with the data (10+20 * 30+400/8) in the key input buffer, the data changes to 660.

The calculation (CALC) key can be used only when the NC control unit is equipped with the custom macro.

ASSIGNMENT SECTION

Section 1

1-1. Why is good attendance especially important for lab courses?

Section 2

2-1. Why would it be unsafe for both the operator and the machine tool if the operator were to use compressed air to blow away chips from the machine tool surfaces?

2-2. Why should chuck jaws be mounted so that they do not protrude too far from the chuck diameter?

Section 3

3-1. What is the value for the maximum swing?

3-2. Describe what swing is.

3-3. What is the maximum horse power for continuous operation?

3-4. What is the value for rapid traverse for the Z axis?

3-5. Explain what constant surface speed is.

3-6. What machine component makes this function possible?

3-7. What is the value for the maximum spindle speed?

3-8. Locate the following items on the machine tool and label the Figure 1-2 with the item number and pointing to its approximate location on the diagram.

24. Paper Tape Reader
25. Tool Unclamp/clamp Button
26. Q-setter Arm
27. Coolant Level Gage

3-9. What type of oil do the slideways require?

Section 4

4-1. Sketch the approximate location of the spindle speed indicator and the load meter on figure 1-6.

4-2. What is the purpose of the load meter?

4-3. What does MDI stand for?

PROJECT SECTION

Competency Evaluation

DATE	DESCRIPTION
_____	State the reason for or explain given safety rules.
_____	Locate specific machine components.
_____	Describe the function of specific control panel features.
_____	Identify proper lubrication points.
_____	Identify CRT control panel features.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 2

Machine Control Unit Operation

INTRODUCTION

Instruction will be given in the manual operation of the machine tool axes and spindle for the purpose of homing or orientating the machine tool. Manual tool loading and turret indexing, and tail stock positioning will also be discussed. Most communication between the operator and the CNC controller will be through a soft key and crt menuing feature which will be introduced in this unit and expanded upon in other units and other courses.

OBJECTIVES

At the completion of this unit the student will be able to:

1. Power up and power down the machine tool.
2. Manually position the machine tool axes.
3. Home the machine tool axes.
4. Manually index the turret and load a tool.
5. Operate the tailstock.
6. Select operations from the soft key and CRT menuing functions.
7. Origin the machine tool axes.

CONTENTS

Reference Section	Page No.
Section 1 Power Up And Power Down	2 -
Section 2 Turret And Tailstock Operation	2 -
Section 3 CRT Menu Operations	2 -
Section 4 Origin The Axis	2 -

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Assignment Section

On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section. 2 -

Project Section

Competency Evaluation 2 -

SECTION 1

POWER UP AND POWER DOWN

Power Up

Follow the steps below to power up or start up the machine tool.

1. Turn on the main breaker of the power control panel located in the back of the machine . The ventilation fan inside the cabinet will turn on and the fluorescent light inside the machine will light up.

2. Turn on the NC side of the control panel by pressing the POWER ON switch. After about 5 seconds the initial screen SYMBOLIC FAPT TURN will be displayed.

3. Reset the EMERGENCY STOP button.

4. Press the STANDBY button.

5. Check the chuck pressure and pump pressure gauges.

6. Press the RESET of the NC unit (FANUC control panel).

7. Press the NC/PC selector key to bring up the numerical control functions of the control panel.

8. Press the function menu key, and then the POSITION soft key to display the ACTUAL POSITION screen.

9. Rotate the MODE SELECT switch to the JOG position.

10. Set the RAPID FEED RATE dial to 79 in/min.

11. Jog both the X and Z axes about 4 inches in the minus (-) direction from the machine zero position. Be certain that during this positioning and the next that there will be no interference with the tailstock.

12. Rotate the MODE SELECT switch to the ZERO RETURN position.

13. Press and hold the X axis toggle switch in the plus (+) direction until the zero return lamp lights up. The turret will travel at the feed rate set in step 10 and then decelerate as it approaches the machine zero point.

14. Press and hold the Z axis toggle switch in the plus (+) direction until the zero return lamp lights up.

15. Set the CENTER JOG FORWARD/RETRACT button to RETRACT and press it for about 5 seconds. Cycle starts or spindle rotation may be inhibited if this is not done.

16. Set the WORK GRIP key switch to the EXT position. This will activate the air chuck to tighten on the outside of the work piece.

17. Press the air chuck foot switch to close the chuck. Cycle starts or spindle rotation may be inhibited if this is not done.

Power Down

The power down or shut down procedures are given below.

1. Home or zero return the X and Z axes as described above.

2. Stop each operation of the machine.

3. Press the EMERGENCY STOP button on the control panel.

4. Press the POWER OFF switch of the NC unit.

5. Turn off the main circuit breaker of the power control cabinet of the machine.

Section 2

TURRET AND TAILSTOCK OPERATION

Indexing The Turret

For manual indexing of the turret:

1. Set the mode switch to HANDLE X, HANDLE Z, JOG or ZERO RETURN (any of the manual modes of operation).

2. Set the TOOL UNCLAMP/CLAMP switch to CLAMP. The turret will not index if it is in the unclamp condition.

3. Check for possible tool interference with other machine components and reposition turret accordingly. Keep your hands and head clear of the turret during the index operation.

4. Press and release the TOOL INDEX button on the control panel. The turret will index one turret station. Note the direction of rotation. By pressing and holding the TOOL INDEX button the turret will swing past several turret stations and stop at the next turret station as soon as the button is released.

The operator must become familiar with the timing for release to become expert at selecting the intended turret station. By pressing and releasing the button successively the turret can be indexed one turret station at a time to get the required turret station at the tool change station.

Tool Change

The unique Quick Tool Change system allows any tool to be changed in a matter of seconds. The tooling is universal so that any turret station will

accept either ID or OD tooling.

To remove and replace a tool:

1. Index the turret station to be changed to the tool change position (one station above the cutting tool or active station).

2. Set the TOOL UNCLAMP/CLAMP button to the unclamp position.

3. Grasp the tool at the tool change position with one hand.

4. Press the TOOL UNCLAMP/CLAMP button. The tool will be released and can be removed.

5. Grasp the tool to be loaded, align the tool with the keyways of the tool pocket and hold in position.

6. Set the TOOL UNCLAMP/CLAMP button to the clamp position.

7. Press the TOOL UNCLAMP/CLAMP button. The tool will be clamped into place. If the tool was not held in place properly then it may not clamp. Press the button to unclamp and then again to clamp (setting the switch position accordingly). If the tool still does not clamp see your instructor.

Tailstock

The tailstock will be covered in more detail in a later unit. Here it is necessary to realize the various modes of operation of the tailstock in order to successfully operate the machine under MDI and program control.

Locate the control panel buttons which operate the tailstock center. The JOG button can be set to FORWARD or RETRACT. Then by pressing the JOG button the center can be moved in or out. The center will move until the

button is released or until the stroke limit is reached. When the JOG button is used to position the center the tailstock pressure is not on when the button is released. Therefore, do not use the JOG button to engage a part with the center since there will be no pressure to secure the part.

The control has a safety feature which will disable the cycle start button if the center is extended and the tailstock pressure is absent. This is why on start up procedures the tailstock JOG RETRACT button is pressed for at least 5 seconds after full retract to ensure built in safety features are set for machine operation.

To properly engage a part with the center and ensure tailstock pressure the FORWARD button must be pressed. It does not have to be held since it will extend until it contacts a part and applies the appropriate tailstock pressure or until it reaches its stroke limit. The green light next to the FORWARD button will then light up signifying tailstock pressure is applied. If this light is not on then the center has not been properly engaged and the cycle start will be disabled.

CAUTION: If the tailstock is too far from the part so that the center is at the stroke limit the green light will be on but the part will not be properly held.

Section 3

CRT MENU OPERATIONS

Soft Key Notation

The 14" MDI & CRT panel is equipped with ten soft keys. The soft keys are displayed as follows:

The characters in the frames are the names of the soft keys displayed at the bottom of the CRT screen.

When some of the operation setting keys (soft keys) or the operation guide keys (soft keys), are pressed, processing is executed immediately.

On the 14" color CRT, they are displayed in green. (These keys are underlined to show that processing is executed immediately after they are pressed.)

The keys in parentheses request input of the data corresponding to the contents of the parentheses. They are also displayed in parentheses on the CRT. When these keys are pressed, the message requesting input of the data is displayed.

When Menu Items Outnumber Soft Keys

As described above, the 14" MDI & CRT panel has the ten soft keys. However, these soft keys are not sufficient for some applications. For example, when the soft keys are used as function selection keys, twelve kinds of the functions can be selected, so ten soft keys are insufficient. In this case, "+" is displayed at the right most end in the lowest line on the CRT.

1. When the soft keys are used as the function selection keys:
   When "+" is being displayed at the right end of the lowest line on the CRT screen, if the function menu key (left most key) is pressed, the soft keys will change over to the function selection keys which have not been displayed so far.

2. When the soft keys are used as the operation selection keys:
   When "+" is being displayed at the right end of the lowest line on the CRT screen, if the operation menu key (right most key) is pressed, the soft keys will change over to the operation selection keys which have not been displayed so far.

3. When the soft keys are used as chapter selection keys:
   When "+" is being displayed at the right end of the lowest line on the CRT screen, if the chapter selection menu key (soft key) is pressed, the soft keys will change over to the chapter selection keys which have not been displayed so far.

Soft Key Changeover Methods

The soft keys are chiefly used as function selection keys, chapter selection keys, operation selection keys and operation guide keys. The soft keys must be changed over to one of the above mentioned keys. The soft keys can be changed over by the procedure described below.

1. Changeover method to function selection keys:

Press the function menu key at the left most end and the soft keys turn to function selection keys.

2. Changeover method to chapter selection keys:

a) Chapter selection in function selection key status:
The function selection key, currently selected, is indicated in green on the MDI & CRT panel. Whenever this function selection key is pressed, the beginning of the next chapter is shown in the CRT.

b) Switching from function selection keys to chapter selection keys:
When the CHAPTER key at the right most is pressed, the function of the soft key is switched from the function selection to the chapter selection. The desired chapter can be selected by pressing the soft key. When bringing the soft keys back to the function selection or the operation selection keys, press the function menu key or operation menu key, respectively.

c) Chapter selection in operation selection key status:
The soft keys are used as the operation selection keys or chapter selection keys to allow the chapter to be selected with the soft keys kept as the operation selection keys. The soft keys are switched to the chapter selection keys by pressing the operation menu key. A new chapter is shown on the CRT when the chapter selection key is pressed. When bringing the soft keys back to the operation selection keys, press the operation menu key.

3. Switching to the operation selection keys:

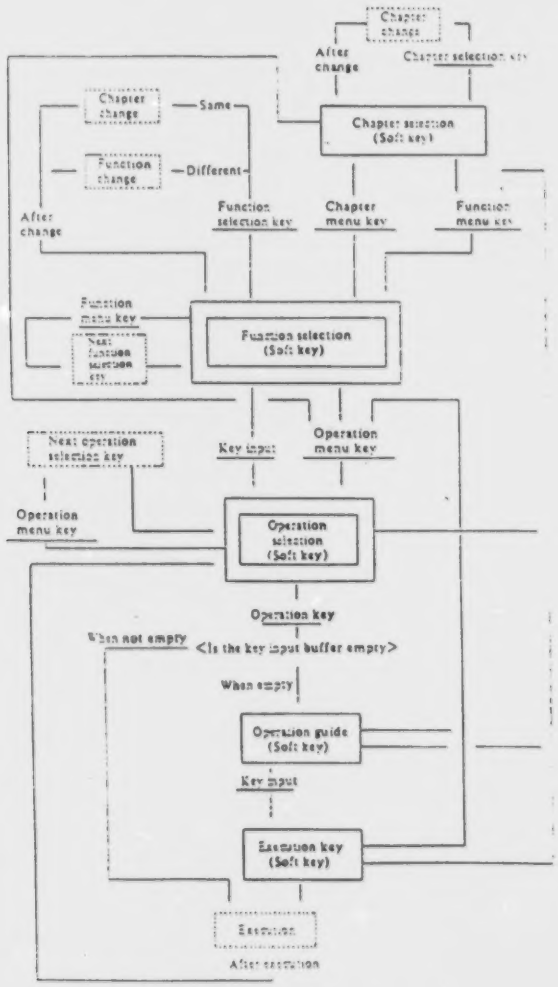
a) When the operation menu key at the right most is pressed, the soft keys are switched to the operation selection keys, irrespective of the current status of the soft keys.

b) When a data is entered to the key-in buffer using the keyboard with the soft keys switched to the function selection keys, the

control unit considers that subsequent operation will be done by the operator and automatically switches the soft keys to the operation selection keys.

4. Changeover method to operation guide keys:
Changeover method from the operation selection keys to the operation guide keys: press an operation selection key.

The block diagram below shows the soft key changeover methods.



BEST COPY AVAILABLE

Summary Of Operations

1. Select the data to be displayed on the MDI & CRT panel, and the major item to be input, with the function selection key (the soft key).

2. Each item is subdivided into smaller items (chapters). Select the small item (chapter) with the chapter selection key (the soft key).

3. To input data, etc. , select the operation with the operation selection key (the soft key).

4. Press the operation selection key. The soft keys are then changed over to the operation guide keys. Possible operations can be determined with the operation guide key.

5. Some operation guide keys start the corresponding operation immediately after they are pressed. Other operation guide keys, when pressed, request the input of the data required for the corresponding operations. The characters of the former keys are displayed in green, and those of latter keys in bluish green.

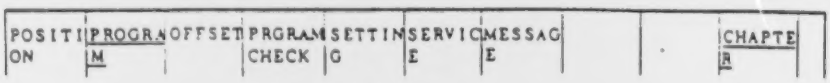
6. When an operation guide key which, when pressed, requests the Input of data required for the corresponding operation, and when the address of the required data has been determined, this address is automatically input to the key input buffer. When the operator must determine the address of the required data, the message requesting the input of the data is displayed on the CRT.

7. When inputting the required data to the key input buffer, the soft key is changed over to the EXEC key which instructs the execution.

8. When the EXEC key is pressed after the required data is input to the key input buffer, the corresponding operation is performed.

Function Selection Keys

To select a function, change over the soft keys to the function selection keys by pressing the function menu key, and press a desired function selection key. The function can be selected in any mode. The detailed function is selected by the chapter selection key. The function selection keys (soft keys) are as follows:



1. POSITION key
   Press this key to select the present position display screen.

2. PROGRAM key
   Press this key to select the part program display screen.

3. OFFSET key
   Press this key to display the tool offset or the work zero point offset display screen.

4. PRGRM CHECK key
   Press this key to display the program check screen.

5. SETTING key
   Press this key to select the setting screen.

6. SERVICE key
   Press this key to display the parameter, diagnosis, etc.

7. MESSAGE key
   Press this key to display the alarm message or operator message.

Chapter Selection Keys

The function selection keys are used to select major items. Each major item is subdivided into smaller items (the chapters). The chapter selection keys are used to select the small items (the chapters). To select the chapter, set the soft keys to the chapter selection keys, and press a required chapter selection key.

1. Chapter selection keys to display position (Function keys to select the chapter of the POSITION) The chapter selection keys to display the position are as follows:

a) OVERALL key
Press this key to select the overall position display screen among the position display screens.

b) RELATIVE key
Press this key to select the relative position display screen among the position display screens.

c) ABSOLUTE key
Press this key to select the position display screen in the work coordinate system among the position display screens.

d) MACHINE key
Press this key to select the position display screen in the machine coordinate system among the position display screens.

2. Program chapter selection keys (Function keys to select the chapter of the PROGRAM) The program chapter selection keys are as follows:

a) TEXT key

Press this key to select the program text screen.

b) DRCTRY MEMORY key

Press this key to display the directory of the programs stored in the program memory. when this key is pressed, the program numbers, program names, etc. are displayed.

3. Offset chapter selection keys (Function keys to select the chapter of the OFFSET) The offset chapter selection keys are as follows:

a) WEAR key

Press this key to select the tool wear offset screen from the offset screens.

b) GEOMETRY key

Press this key to select the tool geometry offset screen from the offset screens.

c) WORK ZERO key

Press this key to select the work zero point offset screen from the offset screens.

4. Command value chapter selection keys (Function keys to select the chapter of the COMMAND) The command value chapter selection keys are as follows:

a) CHECK key

Press this key to display the program currently being executed, the modal values of G codes, F codes, etc. specified by then, the relative coordinate values and residual movement amount. b) LAST key. Press this key to display the command value of the block immediately preceding, the model values of G codes, F codes, etc. specified by then.

c) ACTIVE key

Press this key to display the command value in the block currently being executed, and the modal values of G codes, F codes, etc. specified then.

5. Setting chapter selection keys (Function keys to select the chapter of the SETTING) The setting chapter selection keys are as follows:

a) HANDY key

Press this key to select the setting screen regarding the input/output among setting screens.

b) GENERAL key

Press this key to select the settable parameter screen among the setting screens.

c) OPERAT PANEL key

Press this key to select the software operator panel screen among the setting screens.

d) MACRO VAR. key

Press this key to select the custom macro variable screen among the setting screens.

6. Service chapter selection keys (Function keys to select the SERVICE chapter) The service chapter selection keys are as follows:

a) PARAMETER key
Press this key to select the parameter screen.

b) PITCH ERROR key
Press this key to select the pitch error compensation data screen.

c) DIAGNOSIS key
Press this key to select the diagnosis screen.

d) DISPLAY MEMORY key
Press this key to select the screen that displays the contents of the NC control software memory.

7. Message chapter selection keys (Function keys to select chapter of the MESSAGE) The message chapter selection keys are as follows:

a) ALARM key
Press this key to select the alarm message screen.

b) OPERATOR key
Press this key to select the operator message screen.

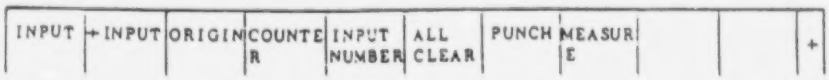
Operation Selection Keys

The possible operation(s) differs according to the screen selected by the function selection key or the chapter selection key. Only the possible operation(s) in the current screen is displayed as the operation selection keys(s) (the soft key(s)), so the operability is improved. To select the operation, set the soft keys to the operation selection keys, and press a required operation selection key.

Note: When the operation selection key is pressed with nothing keyed in to the key input buffer, the soft keys change to the operation guide keys. The operation guide keys will show you operation methods and possible operations. The operation guide keys are convenient for determining unknown operation methods and possible operations. Input the required data to the key input buffer, and press an operation selection key. This data is executed immediately. When the operation method is known without using the operation guide key, this method can reduce the number of key pressing times. However, some operations require the operation guide key. The operation methods using the operation guide keys will be described below.

Example: Offset value setting:

The operation selection keys are as follows:



1. When nothing is keyed in to the key input buffer

   a) Press the INPUT key. The soft keys are changed over to the operation guide key.

b) Press the (VALUE) key. A message to request the key-in of an offset value is displayed.

c) Key in the offset data. The soft keys are changed over as follows:

d) Press the EXEC key. The data in the key input buffer is set in the memory of the offset number where the cursor is located.

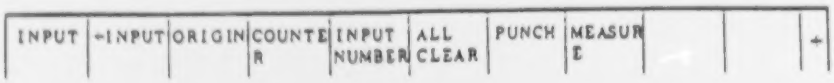
2. When the offset value is input to the key input buffer

a) Press the INPUT key. The data in the key input buffer is set in the memory of the offset number where the cursor is positioned.

Operation Guide Keys

The possible operation(s) differs according to which operation selection key is pressed. Only the possible operation(s) of the pressed operation selection key is displayed as the operation guide key(s) (the soft key(s)), so operability is improved.

Some of the operation guide keys start the corresponding operation immediately when pressed. However, some of them request keying-in of the data required for the start of the corresponding operation. The name of the operation guide key(s) which starts the corresponding operation immediately is displayed in green on the color CRT. In this manual, the notation of these soft key(s) is underlined. When pressing the operation guide key which requests the key-in of the data, a message requesting the key-in of the data is displayed. When keying in the data, the soft keys are changed over as follows, regardless of whether or not the operation guide key is pressed.



When the EXEC key is pressed, the data in the key input buffer is executed.

Key Input And Key Input Buffer

When the address key numerical key is pressed, its corresponding character is input once to the key input buffer. The contents of the key input buffer are displayed at the bottom of the CRT screen. The symbol ">" is displayed in front of it, to indicate that this is the data input by pressing the key(s). At the end of the key input data, "_" is displayed to indicate the input position of the next character.
When the address or the numerical key has two characters, one in the upper and one in the lower section, press the shift key to input the character in the lower section, and then press the address or the numerical key. When the shift key is pressed, indicating the input position of the next character changes to "^", which indicates that the character in the lower section is input (shift status). When one character is input in the shift status, the shift status is cancelled. Therefore, to continuously input the character in the lower section, press the shift key each time the address or the numerical key is pressed. Press the shift key in the shift status to release the shift status. Up to 78 characters at a time can be input to the key input buffer.

Key Input Buffer Contents Change Method

1. Character(s) insertion
   The contents of the key input buffer are displayed on the CRT screen. When the address or the numerical key is pressed, the position where its corresponding character is input (i. e. , the cursor position) is indicated by "_". The cursor position can be shifted by pressing a cursor shift key, -->> or <<--.

Shift the cursor to the position where the character is desired to be inserted, and key in a character desired.

Example: When inserting G90 after N001 of
> N001 X100.0 Z200.0 F100.0;
as the contents of the key input buffer, follow the procedure described below.

- Press the cursor shift key <<-- to shift the cursor to the following position:
>N001_ X100.0 Z200.0 F100.0;

- Key in G90, the display changes as follows:
>N001 G90_ X100.0 Z200.0 F100.0;

- Press the cursor shift key ~ to shift the cursor to the following position:
>N001 G90 X100.0 Z200.0 F100.0;_

2. Character(s) deletion
Shift the cursor to the position next to the character(s) to be deleted, and depress the CAN key.

Example: To delete Z200.0 from
> N001 X100.0 Z200.0 F100.0;_
as the contents of the key input buffer, follow the procedure described below.

-Press the cursor shift key <<-- to shift the cursor to the following position:
> N001 X100.0 Z200.0_ F100.0;

- Press the CAN key 6 times. The display changes as follows:
> N001 X100.0_ F100.0;

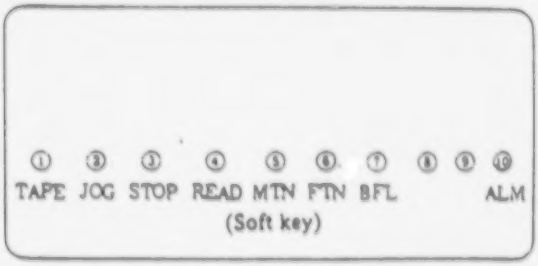
-Press the cursor shift key -->> to shift the cursor to the following position:
> N001 X100.0 F100.0;_

Warning Message

The data must be input in a specified format. When the data is erroneously input to the key input buffer and some operation is executed, a warning message is displayed. On the 14" color CRT, it is displayed in purple on the top line of the key input buffer. When the warning message is displayed, the soft key changes to the cancel key. When the cancel key (the soft key) is pressed, the warning message is cancelled. In this case, the data in the key input buffer can be executed again after the format of the data in the key input buffer is changed to the specified format.

Status Display

This function displays control unit status, whether an alarm is occurring, whether the program is being edited, whether the buffer contains the next block, etc. The second lowest line (immediately above the soft key display line) on the screen is used as a status display line.



The status line is separated into 10 fields (fields I to 10). These fields display the following information (common to all NC screens):

I. Automatic operation mode selection (MEM, MDI, TAPE, EDIT, or ****) This field displays the automatic operation mode being set during display. When automatic operation is not selected, this field displays `*****`.

2. Manual operation mode selection (JOG, ND, INC, AGJ, J + REF, or ****) This field displays the manual operation mode being set during display. When manual operation is not selected, this field displays `*****`.

3. Automatic operation status (RSET, STOP, HOLD, STRT, MSTR, or SRCH). This field displays the status of automatic operation.

   RSET: Reset in progress
   STOP: Automatic operation is stopping.
   HOLD: Automatic operation is in a pause.
   STRT: Automatic operation is being started.
   MSTR: A manual numeric command is being started.
   NSRC: A sequence number is being searched.

4. Program editing status (READ, PNCH, VRFY, SRCH, COND, EDIT, or ****). This field displays the program editing status.

   READ: Registration is in progress.
   PNCH: Punching is in progress.
   VRFY: Collation is in progress.
   SRCH: Search is in progress.
   COND: Memory arrangement is in progress.
   EDIT: Other editing operation (insertion, alteration, etc.) is in progress.
   ****: No editing operation is in progress.

5. Axis movement/dwell status (MIN, DWL, OR ***)
   This field displays `MIN' during axis movement. It displays `DWL' while dwelling is in progress. In other cases, this field displays `***'

6. M, S, T functions status (FIN or ***)
   This field displays `FIN' while an auxiliary function such as M, S. or T is being executed. Otherwise, this field displays `***'.

5/6. Emergency stop status (EMG)
When an emergency stop occurs, the displays 5 & 6 are suppressed; instead, `--EMG--' is displayed, being reversed.

7. Automatic operation buffer status (BFL or ***)
When the automatic operation buffer is not empty, this field displays `BFL'. Otherwise, it displays `***'.

8. Not used.

9. Not used. .

10. Alarm status (ALM BAT or ***)
This field displays "AIM' (reverse flashing) while an alarm is occurring. Otherwise, it displays `***'.

Under the SEICOS LF Conversational Type, when no alarm is occurring, the battery alarm status is displayed. When a battery-out warning signal is generated, this field displays `BAT'. At this time, replace the battery, then reset the system. This field displays `***', again.

Alarm Message Display

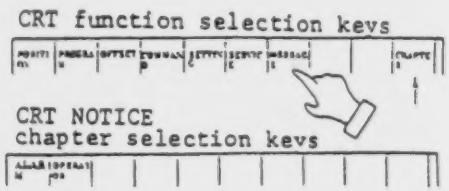
The alarm messages are divided into the following categories:

(i) Alarm messages reported by NC self-diagnosis
(ii) External alarm messages
(iii) Alarm messages reported by custom macro commands

When an alarm message is reported, the alarm message screen is displayed automatically.

1. Press the function menu key to turn the soft key to the operation selection keys.

2. Press MESSAGE key. One of the following screens is displayed.



The alarm message screen is displayed in one of two ways:

i) Press the MESSAGE key repeatedly until the alarm message screen is displayed.

ii) Press the CHAPTER and ALARM keys in this order.

Clearing Alarm Messages

External alarm messages are cleared by external commands. To clear an alarm message reported by NC's self-diagnosis or by a custom macro command, reset the NC unit.

Operator Message Display

The alarm messages are divided into the following categories:

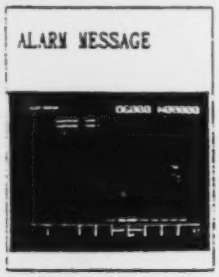
(i) External alarm messages

(ii) Operator messages reported by custom macro commands. When an operator message is reported, the operator message screen is

displayed automatically.

1. Press the function menu key to turn the soft key to the function selection keys.

2. Press MESSAGE key. One of the following screens is displayed.



The operator message screen is displayed in one of two ways:

i) Press the MESSAGE key repeatedly until the alarm message screen is displayed.
ii) Press the CHAPTER and OPERATOR keys in this order.

Clearing Alarm Messages

External operator messages are cleared by external commands. When an operator message reported by a custom macro command is displayed, operation stops as if a single block operation was in effect. When the next cycle is started, this message is erased from the screen.

RESET Key

The RESET key is generally used to clear an alarm. Pressing this key sets the NC control to the following conditions:

Before resetting	After resetting
While executing a travel command	Stops being decelerated. The remainder of the travel command is cancelled.
While sending an M, S, or T code	Stops a sending sequence.
After entering MDI operation data	Cancels all entered data.

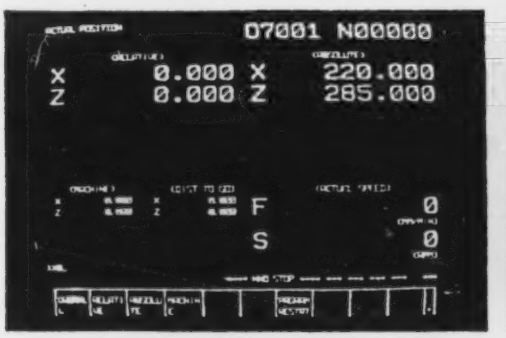
Section 4

ORIGIN THE AXIS

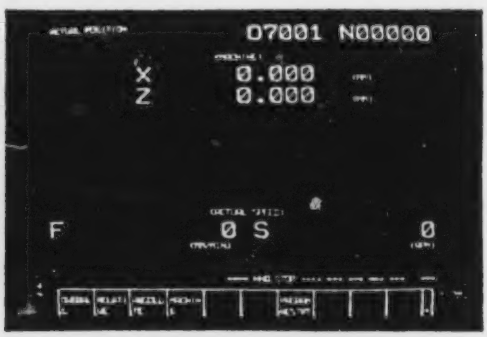
Origin The Relative Position Screen

Typically, after the machine tool has been homed and the X and Z axes are still at
their zero return positions (machine home) the Relative Position screen can be synchronized or aligned with the machine home. This will cause the X0Z0 coordinate value of the relative position screen to correspond to the machine zero return position. This is known as origin the axes.

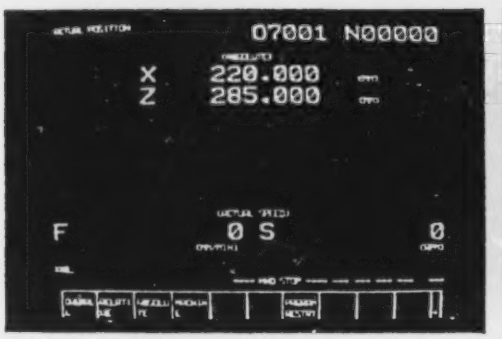
There are four current position display screens:



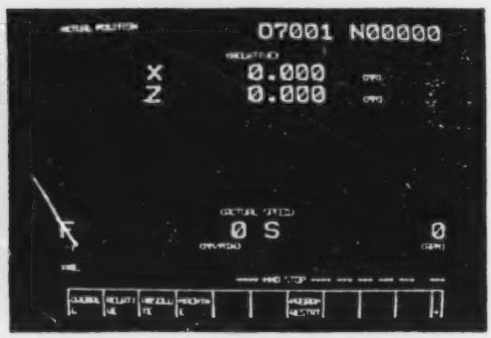
Overall position display



Relative position display



Current position display in the work coordinate system



Current position display in the machine coordinate system

1. Overall position display screen:
   Current position in the relative coordinate system
   Current position in the work coordinate or absolute coordinate system
   Current position in the machine coordinate system
   Residual amount of movement

2. Screen displaying the current position in the relative coordinate system

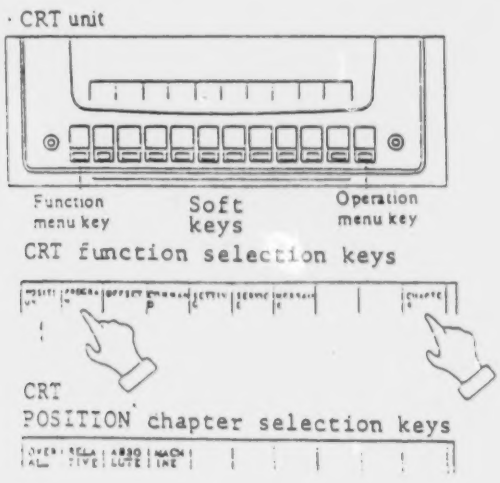
3. Screen displaying the current position in the work coordinate system

4. Screen displaying the current position in the machine coordinate system

Select the current position display screen for the relative coordinate system:

1. Press the Function menu key

2. Press the POSITION soft key

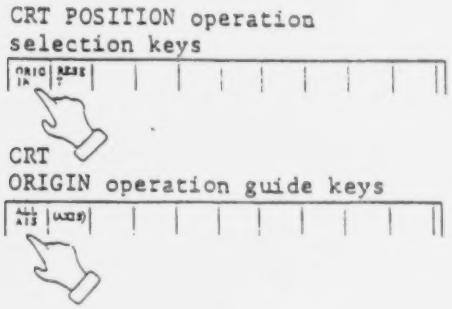


3. Press the CHAPTER soft key (or the Operation menu key)

4. Press the RELATIVE SOFT key

5. Press the ORIGIN soft key

6. Press the ALL AXIS soft key



The X and Z axes coordinate values both read zero now.

Origin The Absolute Position Screen

The origin of the current display screen can be set any where that the turret happens to be positioned at the time that the ORIGIN soft key is pressed. The origin for the absolute position screen can also be changed in this manner. Care should be taken so that the origin is not selected in an arbitrary manner since it is the absolute coordinate system that is used to position the machine tool axes under program control.

Select the current position display screen for the absolute coordinate system:

1. Press the Function menu key

2. Press the POSITION soft key

3. Press the CHAPTER soft key (or the Operation menu key)

4. Press the ABSOLUTE soft key

5. Press the ORIGIN soft key

At this point the only axis which should be originned is the Z axis and usually only if it is at zero return. The X axis absolute position when at zero return is a positive value. This value is the distance between the turret station tool pocket centerline and the spindle centerline.

6. Press the (AXIS) soft key

7. Key in the axis name either, in this case Z

8. Then press the EXEC soft key

If both axes are originned in the absolute position screen then this will effectively shift the work zero to the current position. This has the same effect as the following command when executed under program control:

G50 X0 Z0;

More about program codes in a later unit.

ASSIGNMENT SECTION

Section 1

1-1. Label the first seven steps of the power up procedure in the sequence in which they should be executed.

___ Press the STANDBY button.

___ Press the RESET of the NC unit (FANUC control panel).

___ Reset the EMERGENCY STOP button.

___ Press the NC/PC selector key.

___ Turn on the main breaker of the power control panel.

___ Check the chuck pressure and pump pressure gauges.

___ Turn on the NC side of the control panel by pressing the POWER ON switch.

1-2. Which coordinate direction should each axis be jogged prior to the zero return procedure?

1-3. What position should the MODE SELECT switch be set to perform a machine home procedure?

1-4. What would happen if the axis toggle switch was released prior to the zero return light coming on?

1-5. Refer to 1-4. If the toggle switch was pressed again would the machine home positioning be successful?

1-6. Why must the CENTER JOG RETRACT button be pressed for about 5 seconds after the full retract has occurred?

Section 2

2-1. During the manual tool index, in which direction does the turret rotate?

2-2. If the turret will not move while attempting to perform a manual tool index what are two likely reasons?

2-3. Describe the functions and differences between the center FORWARD button and the JOG FORWARD button.

Section 3

3-1. Where is the function menu key located?

3-2. What is the difference between the function menu key and the operation menu key?

3-3. Start by pressing the function menu key several times, then list each soft key you select to get to the tool wear offset screen.

3-4. What does the HANDY soft key do when it is pressed?

3-5. What does the CAN key do when pressed?

3-6. Look at the status display after the machine tool has been homed. List the control unit status indicated at each field and describe what this indicates?

3-7. What happens if the RESET key is pressed during a zero return procedure?

Section 4

4-1. Where is the turret usually positioned when the relative position screen is originned?

4-2. Which axis is usually not originned when the absolute display screen is originned?

4-3. When the X axis is zero-returned what is the distance between the turret station tool pocket centerline and the spindle centerline?

PROJECT SECTION

Competency Evaluation

DATE	DESCRIPTION
_____	Power up and power down the machine tool
_____	Home the turning center
_____	Perform a tool change
_____	Operate the tailstock
_____	Origin the relative position screen

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 3

MDI And The Edit Mode

INTRODUCTION

This unit will prepare the operator to execute short programs created through the use of the Manual Data Input (MDI) mode of the control panel. MDI can be used for many purposes from simply setting initial operating parameters to setting spindle rotations for warm up purposes to machining soft jaws or rough machining work pieces. The edit mode will also be introduced here for the purpose of loading and editing CNC part programs.

OBJECTIVES

At the completion of this unit the student will be able to:

1. Interpret CNC machine code
2. Utilize MDI to index the turret, set spindle speeds, and position the machine tool axes.
4. Load a part program from the control memory.
5. Edit a part program.

CONTENTS

Reference Section	Page No.
Section 1 CNC Machine Code	3 -
Section 2 Manual Data Input Mode	3 -
Section 3 Edit Mode	3 -

Assignment Section

On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section.

Project Section

Competency Evaluation 3 -

SECTION 1

CNC MACHINE CODE

Preparatory Commands

Preparatory commands are G codes which the CNC machine tool interprets in order to perform specific machining functions such as positioning system selection, interpolated tool path axis synchronization, and canned cycle selection among other functions.

G CODES			
Group	**G Code**	**Function**	
01	G00	Positioning	•
	G01	Linear interpolation	•
	G02	Circular interpolation CW	•
	G03	Circular interpolation CCW	•
00	G04	Dwell	•
	G09	Exact stop	
	G10	Data setting	
	G10.1	PC data setting	
	G11	Data setting mode cancel	
06	G20	Inch input	
	G21	Metric input	
04	G22	Stored stroke check on	
	G23	Stored stroke check off	
00	G27	Reference point return check	
	G28	Return to 1st reference point	•
	G29	Return from reference point	
	G30	Return to 2nd reference point	
	G31	Skip function	

Group	G Code	Function
01	G32	Thread cutting
	G34	Variable lead thread cutting
	G36	Automatic tool offset (X axis)
00	G37	Automatic tool offset (Z axis)
	G37.1	Automatic tool offset #1
	G37.2	Automatic tool offset #2
	G37.3	Automatic tool offset #3
07	■ G40	Tool nose radius compensation cancel
	G41	Tool nose radius compensation left
	G42	Tool nose radius compensation right
00	G50	Programming of absolute zero point/ spindle maximum speed setting •
00	G52	Local coordinate system setting
	G53	Machine coordinate system selection
14	■ G54	Work coordinate system 1 selection
	G55	Work coordinate system 2 selection
	G56	Work coordinate system 3 selection
	G57	Work coordinate system 4 selection
	G58	Work coordinate system 5 selection
	G59	Work coordinate system 6 selection
15	G61	Exact stop mode
	■ G64	Cutting mode
00	G65	Macro call
13	G66	Macro modal call A
	G66.1	Macro modal call B
	■ G67	Macro modal call A/B cancel
11	G68	Mirror Image for double turrets on
	G69	Mirror Image for double turrets cancel

Group	G Code	Function	
00	G72	Finishing cycle	
	G73	Stock removal in turning	
	G74	Stock removal in facing	
	G75	Pattern repeating	
	G76	Peck drilling in Z axis	
	G77	Grooving in X axis	
	G78	Threading cycle	
01	G90	OD/ID turning cycle	
	G92	Canned cycle for thread cutting	•
	G94	End face turning cycle	
12	G96	Constant surface speed control	•
	■ G97	Constant surface speed control cancel	•
05	G98	Feed per minute	
	■ G99	Feed per revolution	

1. The G codes marked with ■ are initial G codes in each group. That is, these G codes are set when the power is turned on or when the RESET button is pressed. For G22 and G23, G22 is selected when the power is turned on. After reset, G22 or G23 (the one effective before resetting) is set. For G20 and G21, the one in effect before the power was cut or the RESET button was pressed is selected.
2. When conventional programming is done, all the G codes listed can be used. When programming is done utilizing the symbolic FAPT system, only the G codes marked can be used.

Miscellaneous Functions

Miscellaneous functions are M codes which the CNC machine tool interprets in order to control machine tool functions which are independent of tool path positioning such as coolant control, spindle control, tailstock and chuck operation among other functions.

M CODES

M Code	Function	M Code	Function
M00	Program stop	*M30	Program end
*M01	Optional stop	M31	No-blank detection
M02	Program end	*M40	Spindle low-speed range
*M03	Spindle forward start	*M41	Spindle high-speed range
M04	Spindle reverse start	M48	Feed rate override cancel OFF
*M05	Spindle stop	M49	Feed rate override cancel ON
*M08	Coolant ON	M61	Auto door open
*M09	Coolant OFF	M62	Auto door close
M12	Work count	M63	Unloader OUT
M21	Error detect ON	M64	Unloader IN
M22	Error detect OFF	M66	Chuck clamp pressure low
M23	Chamfering ON	M67	Chuck clamp pressure high
M24	Chamfering OFF	M68	Chuck close
M25	Tailstock spindle forward	M69	Chuck open
M26	Tailstock spindle retract	M98	Subprogram call
M27	Tailstock forward	M99	End of subprogram
M28	Tailstock retract		

SECTION 2

MANUAL DATA INPUT (MDI) MODE OF OPERATION

MDI Operations

MDI operations are performed by the operator primarily in performing set up operations or preparations for machining or testing machine functions.

MDI operations can only be performed when the mode selector switch is set to the MDI position. When this is done the program that was displayed on the CRT is removed although the program number is still displayed in the upper right of the CRT. The upper left of the CRT displays the selected mode of operation which states "PROGRAM (MDI)".

The MDI commands are entered from the keyboard and displayed in the lower left of the CRT. Then these commands are inserted into the MDI operation buffer memory by using the INSERT soft key. These commands are now displayed in the upper left corner of the CRT.

MDI Operating Procedures

The following sequence of procedures reveal to the operator the status of the machine tool as far as active G codes, M codes and offsets. If certain G codes are not active and are required to perform an MDI operation the operator must program these in the MDI mode. For example if the turret needs to be moved in an incremental mode of positioning then the G91 code must be active. If it is not active then the operator must include this G code as part of the MDI program.

1. Set the mode selector switch to the MEMORY position.

2. Press the function menu key to display the main menu.

3. Press the PRGRAM CHECK soft key to display the PROGRAM CHECK screen. Verify the currently active G codes, and M codes.

4. Press the function menu key to display the main menu.

5. Press the PROGRAM soft key.

6. Set the mode selector switch to MDI.

7. Use the keyboard to input the data required.

8. Press the INSERT soft key.

9. Press the function menu key.

10. Press the PRGRAM CHECK soft key.

11. Press the CYCLE START or START button to execute the MDI program instructions.

The MDI program instructions are removed from the MDI memory buffer as soon as they are executed. An MDI program can only be run once, then it has to be reentered in order to execute it again.

Index The Turret With MDI

1. Home the machine tool.

2. Check for any possible interference between any tool on the turret and the workpiece or tailstock that might occur during a turret index.

3. Call up the MDI operation mode by the procedures outlined above.

4. Key in the following:
   T0200;
   The semicolon is the symbol for the end of block character that the FANUC control uses. To key in this symbol first press the shift key, then press the key with the letters: BS $_{EOB}$

5. Press the INSERT soft key.

6. Make sure hands and head are clear of the turret, press the CYCLE START button. Note the direction of rotation of the turret as it indexes.

7. Repeat steps 4, 5, and 6 for each of the following turret positions in the order given: T0400; T0100; T0600; T0100;

Operate The Spindle With MDI And Manual Modes

The control panel has several buttons to manually operate the spindle, which are on, stop, jog and rotation direction, but there is no dial to set the spindle speed. When the SPINDLE JOG or SPINDLE ON button are pressed the spindle rotates at the speed last programmed in the MDI statements. Therefore the operator must program the desired spindle speed in the MDI mode to control the spindle in the manual mode of operation.

1. Call up the PROGRAM(MDI) display as outlined above.

2. Key in the following :
   G97S1000;
   The G97 commands the machine to rotate the spindle at constant speed.

3. Press the INSERT soft key.

4. Press the CYCLE START button. Nothing should happen yet unless an M03 or M04 miscellaneous code (M code) is active. If one of these codes was active the spindle may not begin rotating for reasons which will be stated next.

5. Set the mode selector switch to any of the manual modes.

6. Activate the chuck with the foot switch to close the chuck.

7. Close the chip shield door until it makes the safety switch at the top door rail.

8. Set SPINDLE FORWARD/REVERSE switch to FORWARD.

9. Set the SPINDLE START/JOG switch to START. Press this button.

10. The spindle should now rotate at the programmed speed. If it does not start check that the chuck is closed, that the chip shield door is closed, and that the center has been fully retracted by pressing the CENTER JOG RETRACT button for 5 seconds after it has retracted fully. If it still does not start call your instructor.

11. The door can now be opened once the spindle begins to rotate.

12. Press the SPINDLE STOP button.

The conditions specified in item 10 above must be met each time the spindle is started. The SPINDLE JOG selection can be pressed and the spindle will rotate at the programmed speed as long as the button is held in. The door can be opened or closed, but the chuck must be closed.

Operate The Spindle With MDI

The spindle can also be controlled entirely with the MDI program commands.

1. Call up the PROGRAM(MDI) display as outlined above.

2. Key in the following :
   G50S1500M41;
   G96S100M03;
   When keying in multiple lines of code the INSERT soft key can be pressed after each EOB character or at the end of the last EOB character. Don't press the START button until after all code has been INSERTED! The G50 limits the spindle speed to a maximum 1500 rpm given by the S1500. The M41 commands the high speed spindle operation.The G96 commands the machine to rotate the spindle at constant surface speed. In this line the S word signifies 100 surface feet per minute, the velocity that the tool point will move relative to the workpiece surface. This velocity will be held constant as the tool repositions to different diameters up to the 1500 rpm limit. As the tool moves in or out along the x axis the spindle rpms will increase and decrease. The M03 specifies a clockwise rotation of the spindle.

3. Press the INSERT soft key.

4. Press the CYCLE START button. The spindle should begin rotating.

5. Key in the following:
   M05;M01;M04;M01;M05;M01;M03;
   The M01 is an optional stop and will be used to step through this program.

6. Set the OPTIONAL STOP toggle switch to the ON position.

7. Press the CYCLE START button. After each optional stop is executed note what the spindle is doing, then press the CYCLE START button. Do this until all code has been executed. The spindle should be rotating when completed.

8. Set the mode selector switch to the X HANDLE manual mode.

9. Set the move multiplier switch to the 50/1 position.

10. Rotate the PULSE GENERATOR dial in the minus direction. As the turret moves closer to the spindle center the rpms change as indicated on the spindle speed meter. Note how the direction of the turret movement affects the rpms.

11. Press the SPINDLE STOP button.

SECTION 3

EDIT MODE

Selecting A Program From Directory Memory

As an operator many of the part programs you will setup and run on the turning center will be loaded into the directory memory of the CNC controller. These programs are loaded into the controller via a paper tape reader which may be integral to the machine tool or connected to an external communication port. The programs can also be loaded via DNC (distributed numerical control) which uses an internal communication port connected to another computer system where the CNC part programs are stored and/or developed. See your instructor for a demonstration on how your machine tool is loaded with the part program.

Once the turning center has been homed follow the procedures listed below to view and select a program from the directory memory.

1. Set the mode switch to MEMORY.

2. Press the function menu key.

3. Press the PROGRAM soft key.

4. If the program text does not appear on the display then press the CHAPTER soft key, then press the TEXT soft key. The current program text will now be displayed.

5. If the program displayed does not appear to be at its start or beginning, (the first line should be the program number beginning with the letter O), then press the operation menu key, and finally the REWIND soft key.

6. If the program now displayed is the one you require then you are ready. If not then you must look at the directory memory to see if the program you want is loaded into the controller. You should still have the PROGRAM menus selected. Press the operation key until the words DIRECTORY MEMORY appear in the menu. Press the DIRECTORY MEMORY soft key. The program text will disappear and in its place a listing of the part programs will appear in the display.

7. If the program required is not listed see your instructor. Otherwise key in the program number. This number should begin with the letter O followed by four numbers. It will be displayed in the lower left hand corner of the display.

8. Press the operation menu key until you see the abbreviated words FRWRD SEARCH and BKWRD SEARCH in the menu.

9. Press either of the FRWRD or BKWRD SEARCH soft keys. The new program will now become current. Use the operation key to look for the TEXT soft key and press it to see the program text.

Practice loading in different programs from the directory and use the cursor keys to look at the different part programs.

The MEMORY mode is the mode from which the part program is executed and machining under program control is accomplished. During the setup and prove out of the part program it may become apparent that the program will need to be modified in order to correctly machine the part. This will require editing of the program and will use the EDIT mode of the mode select switch. Occasionally the controller memory will have insufficient space to load in another program into its directory memory and will require some other programs to be removed or deleted from its memory. This is also the function of the EDIT mode.

Program Deletion

Locate the MEMORY LOCK/WRITE key on the control panel. When this key is turned to the LOCK position then it will be impossible to alter or delete part programs in the controller memory. To begin any editing activity you must turn this key to the WRITE position.

1. Turn the MEMORY LOCK/WRITE key to the WRITE position.

2. Select the EDIT mode position.

3. Press the operation menu key until the word DELETE appears in the menu.

4. Press the DELETE soft key.

5. Press the PROGRAM soft key.

6. Carefully make your next menu selection. Do not press the ALL soft key at this time. Pressing the ALL soft key will delete all programs from memory, (O8000 and O9000 series programs are special and have additional protection so they will not be deleted). Do not press the THIS soft key. Pressing the THIS soft key will delete the current selected part program. It is recommended that you press the PROG# soft key. This will allow you to explicitly tell the controller which program to delete.

7. Input the program number you wish to delete. Or enter several program numbers separated by commas.

8. Press the EXEC soft key. The programs are now deleted.

Verify the program deletions by calling up the directory memory.

Program Editing

The contents of a program stored in memory can be edited or modified from the controller. There are a variety of ways in which to locate the program text that requires modification and then a variety of ways in which to perform the modification. The following general procedure applies. The procedural details will follow.

1. Load the program to be edited into the current memory. Follow the instructions provided in "Selecting A Program From Directory Memory".

2. Set the mode switch to EDIT. Turn the MEMORY LOCK/WRITE key to WRITE.

3. Move the cursor to the program text to be modified.

   3-1. Press the operation key.

   3-2. Press either the FRWRD SEARCH or BKWRD SEARCH soft key. Press the FRWRD SEARCH key if the text to be modified is toward the end of the program relative to the current position of the cursor.

   3-3. If you pressed the FRWRD SEARCH soft key you can now press the BOTTOM soft key which will position the cursor at the program end, or you can press the WORD soft key.

   3-4. If you pressed the WORD soft key then key in or enter the address (for example N, or X), or the word (N100, X2.75).

   3-5. Press the EXEC soft key. The address or word will then be searched for in the direction you picked from the current position of the cursor. The cursor will highlight the first word it finds.

4. Perform the modification to the text. The word highlighted can now be altered, have text inserted after it, or be deleted.

Alteration of Text

4-1. Press the operation key.

4-2. Press the ALTER soft key.

4-3. Press the WORD soft key.

4-4. Enter the text required. It can be a single word, several words, or several blocks.

4-5. Press the EXEC soft key. The highlighted word will be replaced with the entered text.

Insertion of Text

4-1. Press the operation key.

4-2. Press the INSERT soft key.

4-3. Press the WORD soft key.

4-4. Enter the text required. It can be a single word, several words, or several blocks.

4-5. Press the EXEC soft key. The entered text will be inserted immediately after the word highlighted by the cursor.

Deletion of Text

4-1. Press the operation key.

4-2. Press the DELETE WORD soft key. The highlighted word will be deleted.

Or...

4-1. Press the DELETE soft key.

4-2. Press the WORD soft key. The highlighted word will be deleted.
Or...
Press the ~EOB soft key. Everything from the highlighted word to the next EOB (~) will be deleted.

Or...

4-1. If you wish to delete several blocks: Position the cursor on the first word of the block you are going to start deleting.

4-2. Press the DELETE soft key.

4-3. Press the ~ WORD soft key.

4-4. Enter the word which is the last word of the series of blocks you wish to delete.

4-5. Press the EXEC soft key. All the text from the highlighted word to the last entered word will be deleted.

An example program will be provided to you to practice the various methods of editing. Any programs you practice on should be verified for correctness prior to any execution of machining.

ASSIGNMENT SECTION

Section 1

1-1. What do preparatory commands do?

1-2. What do the following codes do?

G00
G01
G03
G50
G96
G97
G99

1-3. What do miscellaneous commands do?

1-4. What do the following codes do?

M00
M01
M03
M05
M26
M30

Section 2

2-1. What does MDI stand for?

2-2. Before performing MDI operations why must you display the PROGRAM CHECK screen?

2-3. Write a short program for MDI that will index the turret to position 8, turn on the spindle clockwise at 1200 RPM, dwell for 10 seconds, then stop the spindle, and index the turret to position 5.

Section 3

3-1. What display screen will show you the part programs that are stored in the controller?

3-2. What is the soft key menu sequence required to show the screen from 3-1 above and then select a program to be made current?

3-4. When using the INSERT soft key is the entered text placed in front of or behind the highlighted word?

PROJECT SECTION

Competency Evaluation

DATE	DESCRIPTION
_____	Enter and execute the MDI program you created in the assignment section.
_____	Select a program from directory memory and make it current.
_____	Demonstrate specific editing procedures on the current program.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 4

Setup Methods

INTRODUCTION

This course will introduce the student to one of several methods of setting up the CNC turning center machine tool. Program verification or prove out procedures will also be introduced in this unit. These include tape check, dry run, machine lock, and trial cut.

OBJECTIVES

At the completion of this unit the student will be able to:

1. Set the work zero coordinate.
2. Qualify the tool point of O.D. and I.D. tooling using the tool preset method (G50)
3. Perform a program tape check, with and with out machine lock .
4. Perform a dry run.
5. Perform a trial cut.

CONTENTS

Reference Section	Page No.
Section 1 Turning Center Setup - G50 Method	4 -
Section 2 CNC Program Prove Out	4 -

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Assignment Section

On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section.

Project Section

Demonstration Project	4 -
Competency Evaluation	4 -

SECTION 1

TURNING CENTER SETUP - G50 METHOD

CNC Documentation

The CNC documentation is a job packet that will be issued to the machine operator or stored in a file cabinet near the machining center. The documentation consists of all the information and instructions required to properly setup the machine tool including: the part drawing, the CNC part program, the setup plan, the tool list and operation sheet, and the offset work sheet. Examples of these documents are in the project section of this unit. Please refer to them when answering the unit questions.

You, the operator, have the responsibility of verifying the tooling called out in the tool list for: proper assembly down to the tool nose radius specified on the insert, correct position in the turret, and proper extension of tooling when special instructions call for clearance or reach.

The setup plan must be followed closely and double checked for errors that the programmer might have committed or for machine tool variations that could cause problems.

Finally the part program print out or hard copy must be compared line for line with the program in the control. If there are any differences found notify your instructor (or supervisor). Make corrections to the program only after you determine which version is incorrect. Also verify that the tooling positions in the turret called out in the program match the actual tooling positions in the turret. Check the cutting speeds and feed rates in the program with the cutting speeds and feed rates specified in the tool list.

Premachining Parts

Occasionally you may be required to do some manual premachining of parts to qualify the lengths or chucking diameters. This may be a routed operation requiring a separate setup plan or it may be special instructions included with the setup plan for the CNC program. If special instructions are not provided always study the job and make sure it can be run as is or consult your instructor (supervisor) if you are not sure. Sometimes soft jaws will have to be premachined if chucking on a finished diameter. This will be discussed later in this unit.

The project NC-332.0 which we will be referring to will be demonstrated by your instructor as we proceed with the setup procedures and discussion. The tooling for this project will be assembled and loaded by your instructor. This project will require three premachining operations:

1) face, turn and chamfer the left end (3.000 x 2.500 dia.)
2) face and center drill the right end
3) machine 2.500 dia. soft jaws

Review the print of NC-332.0 at the end of this unit to see if you agree with the above statement. Your instructor will perform the premachining operations as part of the setup demonstrations for this unit.

Hard Jaws

There are several ways of holding the workpiece in the spindle of the machine. These include: the 3 jaw chuck, which we will be using extensively; the 4 jaw chuck, the collet chuck, and special holding fixtures for castings, prismatic or flat parts.

On the first two premachining operations we will be using the 3 jaw chuck with hard jaws. The hard jaws must be assembled to the chuck so that they do not protrude more than 1/4" beyond the outside diameter of the

chuck. Make sure the hard jaw numbers match the chuck master jaw numbers. Count or mark the serrations on the back of the hard jaws so that all the jaws will be mounted in the same position.

Carefully feed the rough stock into the chuck and beyond into the spindle stock tube so that the stock face is the prescribed distance from the hard jaw face. This information should be included in the setup plan. Set the WORK GRIP key to EXT. Close the chuck. Turn on the spindle and see if the job runs true.

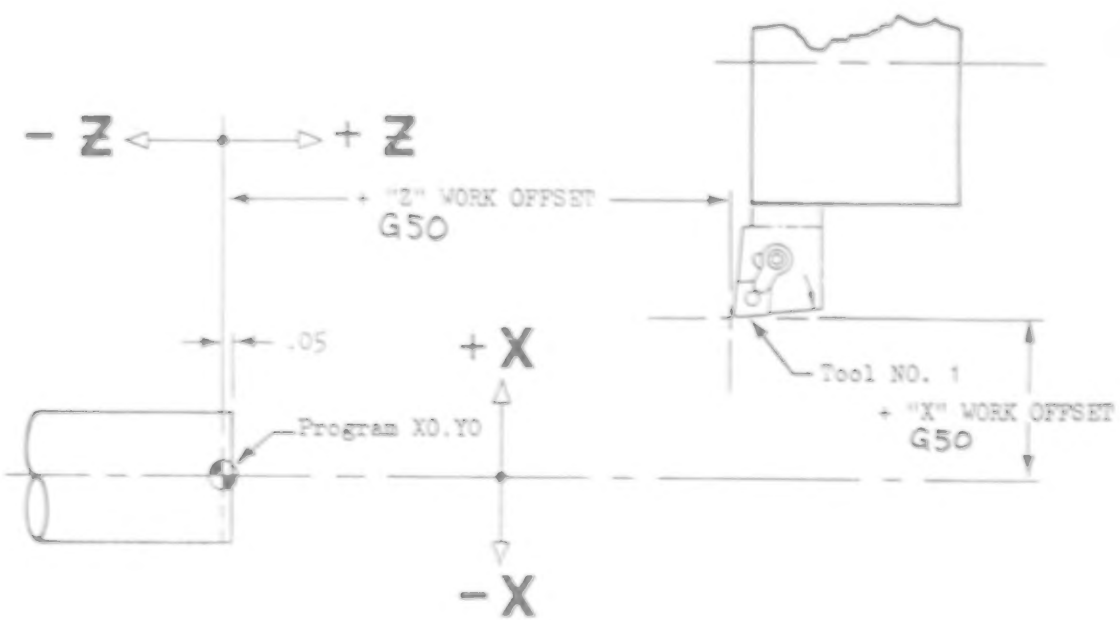
Manual Operations And The G50 Tool Point Definition

The premachining operations are usually performed manually with either the jog feed controls or the handle control if the tool positioning is uncomplicated and the tool cut path is relatively short. Another preferred way is programming the tool path and spindle speeds in the MDI mode. Regardless of which method is chosen it is recommended that the tool point location be defined with the G50 method especially since this is the method we will be concentrating on in this text.

The G50 setup method will be described shortly when we discuss the setup of the programmed machining operation. When used in either the manual or MDI mode the tool point location for the current tool relative to the workpiece face (which is referred to as work zero or Z0) is defined in a G50 block with the X and Z words. The G50 block must be executed in the MDI mode for each tool as it is called up (for example after the turret is indexed to another tool). Caution must be taken when either a turret index occurs or the control is RESET as the G50 will no longer apply to the current tool or may be cancelled. When executing any position moves always refer to the distance to go registers on the CRT and visually corroborate the tool position relative to the work piece.

The G50 Tool Point Definition

Once the workpiece has been properly located and chucked the work offset of the part must be established for each tool. Since each tool has different geometries or is assembled in the tool holders in slightly different positions the relative location of the tool point to the work zero will be different for each tool. This distance or work offset will be established from the machine home or zero return position of the turret. See the figure below. The correct G50 values must be edited into the program for each tool. When using this method you must make certain that the program performs all turret indexes and G50 definitions at the zero return position. The procedure follows.



1. Zero return the machine in both X and Z.

2. Clear all wear offsets and all geometry offsets.

   2-1. Press the function menu key.

   2-2. Press the OFFSET soft key.

   2-3. Press the operation menu key.

   2-4. Press the ALL CLEAR soft key.

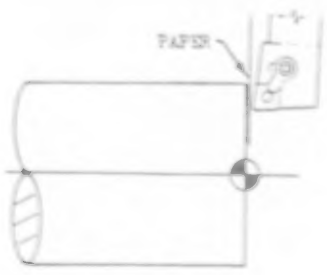
   2-5. Press the ALL soft key.

3. Obtain the OFFSET WORKSHEET from your job packet. Use this worksheet to calculate and record the work offset to be used in the program G50 block.

4. Display the ACTUAL POSITION screen on the CRT. Verify that the (Machine) position display is zero for both X and Z values.

5. Manually index the turret to the tool required. For this example will we assume the first tool is an O.D. tool (face and turn).

6. Jog the tool in Z and X to about one quarter to one half inch from the face of the workpiece. Use the handle to make a final approach to within a thirtysecondth of an inch.



7. Set the MOVE MULTIPLIER switch to 1/1 and touch off the face of the part using a paper feeler to protect the tool insert. As shown above.

8. Look at the (Machine) position display. The Z axis position should be a negative value. This value is the distance the tool point traveled from zero return to the workpiece face (which we will also call work zero, program zero, or Z0). The absolute value of this machine Z position is what we will use to calculate the G50. The G50 Z word value is equal to the absolute value of the machine Z position plus the amount of finishing allowance left on the workpiece face. The amount of finishing allowance should be noted on the setup sheet.

   G50 Zvalue = |machine Zvalue| + f.a.

9. Record this result on the OFFSET WORKSHEET in the Z column under the Tool Length Offset -Geometry/G50 section.

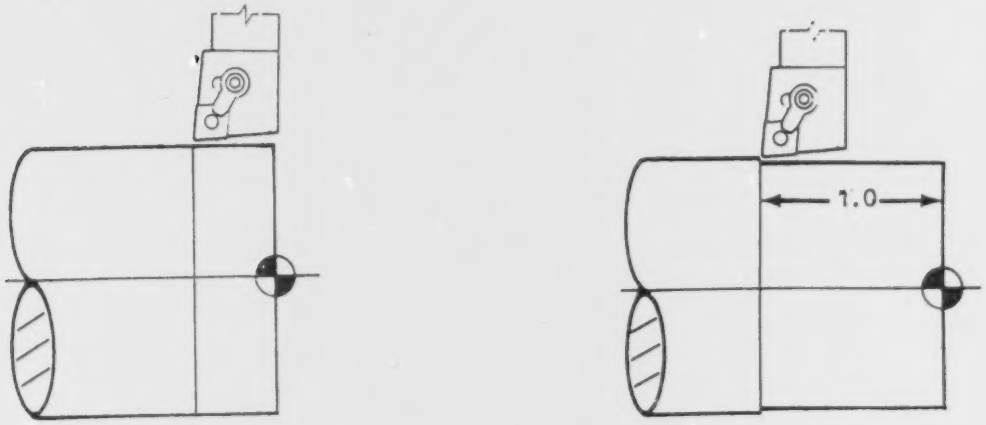
10. Back the tool away from the face of the part in the +Z direction. Use the jog mode and the handle mode to reposition the tool point in the X axis and touch off the outside diameter (O.D.) of the part.

11. Look at the (Machine) position display. The X axis position should be a negative value. This value is the distance the tool point traveled from zero return to the workpiece O.D. in units of diameter equivalent. The absolute value of this machine X position is what we will use to calculate the G50. The G50 X word value is equal to the absolute value of the machine X position plus the outside diameter of the workpiece that the tool point is resting on. This diameter is found by actually measuring it with a micrometer.

    G50 Xvalue = |machine Xvalue| + O.D.

    If this is not a finished diameter and you suspect it may not be concentric with the spindle centerline then you should turn the O.D. with a skin cut to just clean up the O.D. Record the machine X value

for this O.D. cut and then mic the O.D. and add this amount to the machine X value.



12. Record this result on the OFFSET WORKSHEET in the X column under the Tool Length Offset -Geometry/G50 section.

13. Back the tool away from the O.D. of the part in the +X direction. Use the jog mode to reposition the tool point so that it will clear the part and tailstock.

14. Zero return the machine to prepare for a turret index.

15. Repeat steps 5 through 14 for each tool with the following considerations for lefthand tools and I.D. tools.

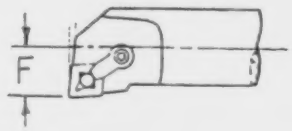
All drills will have the same G50 Xvalue and that is equal to 17.000. Perform the face touch off procedure above to calculate the G50 Zvalue.

G50 Xvalue = 17.000 (for all drills, reamers, and taps)

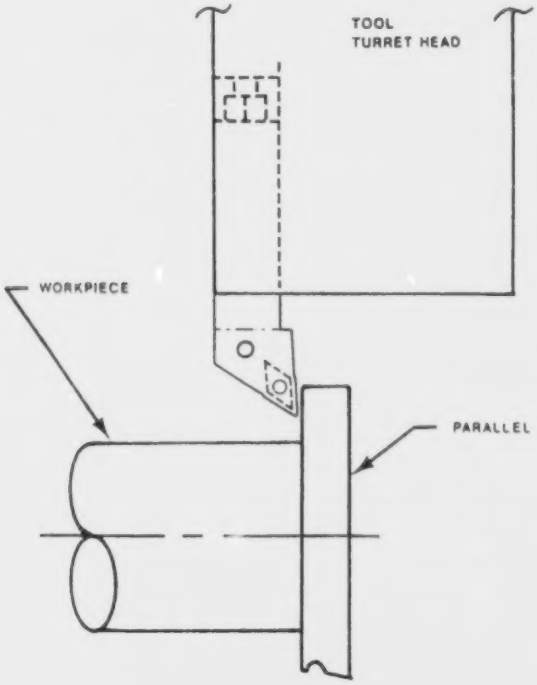
All boring bars should point inward toward the turret centerline. This effectively will make their G50 Xvalues all greater than 17.000. This is

calculated by adding twice their associated F dimension to 17.000. The F dimension is the distance from the boring bar centerline to the boring bar tool point. This distance is the radius of the hole the boring bar would make if it were rotated about its own centerline. Twice this distance is the diameter of the hole and it is the diameter units we need to add to the machine position to get the correct G50 Xvalue. See the example below. Perform the face touch off procedure above to calculate the G50 Zvalue.

G50 Xvalue = 17.000 + (2*F) (for all boring bars)



Lefthand tools should use a parallel as shown below to touch off the face of the workpiece. Otherwise, perform the touch off procedures outlined above.



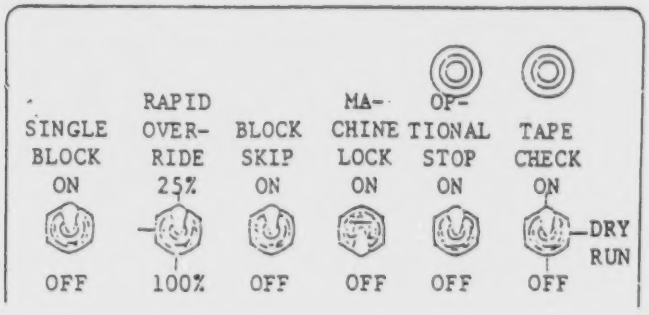
16. Carefully review the program for every tool and its G50 block. Edit the program so that the G50 X and Z word values match the calculated values you recorded in the OFFSET WORKSHEET.

You are now ready to begin the program verification procedures known as program prove out. On this demonstration project your instructor will be demonstrating the procedures that follow. Use these procedures on all the upcoming projects that you will be machining.

SECTION 2

CNC PROGRAM PROVE OUT

There are a series of program executions that will be performed in order to safely check the correctness of the CNC program you are about to run. These different program executions are dependent on the status of several toggle switches which can be applied independently or in conjunction with one another. The different runs include tape check, dry run, trial cut, and finally full automatic.



The chart on the following page will describe their specific function.

Switch	Position	Function
SINGLE BLOCK	ON	The machine stops its operation after executing one block. Pressing the START button proceeds to execution of the next block.
	OFF	Continuous operation
RAPID OVERRIDE	100%	┐
	25%	├ Select a safety speed.
	5%	┘
BLOCK SKIP	ON	A block containing "/"(slash) is skipped.
	OFF	A block containing "/"(slash) is executed.
MACHINE LOCK	ON	The turret head stops at a present position and does not move. The rest of the machine functions.
	OFF	The turret head functions as commanded.
⊖ OPTIONAL STOP	ON	M01 function is executed (lamp lit up). To restart, press the START button.
	OFF	Operation proceeds to the next block, skipping M01.
⊖ DRY RUN	TAPE CHECK	Spindle rotation is suspended in addition to the dry run function.
	DRY RUN	A manually set feed rate becomes effective, ignoring rapid traverse and cutting feed on the tape.
	OFF	The tool travels as commanded.

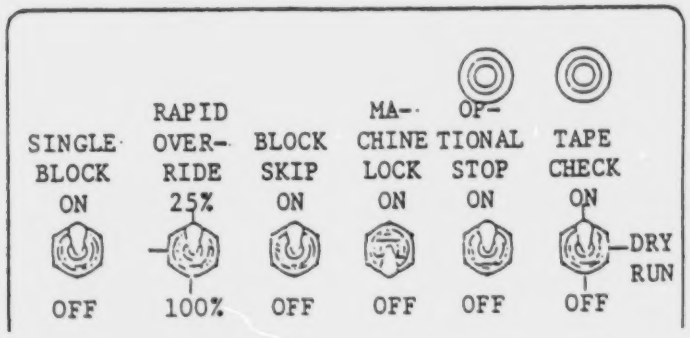
Tape Check

Tape check with machine lock on essentially freezes all mechanical motion of the machine tool. The program is executed logically as if it were controlling the machine normally. This check will catch format errors and syntax errors that are in the program. These errors usually result from typing mistakes and will cause program execution to halt. This then requires the time consuming task of clearing the tool point from the part, resetting the program, and zero returning the machine.

Whenever a new program has been loaded into the control or extensive editing has been done to the program it should be run with both tape check and machine lock on.

Tape check with out machine lock on will allow tool travel to occur with no spindle rotation. In this manner tool travel, tool over travel and tool interference as well as program contents can be checked.

1. Place the toggle switches in the position shown below.



2. Zero return the turret in both X and Z.

3. Open the chuck, remove the workpiece, and close the chuck.

4. Position the RAPID FEED RATE dial to 79 in/min. This selects the feed for both the rapid position moves and all the feed rate position moves.

5. Set the mode switch to MEMORY.

6. Reset the control and Press the REWIND soft key.

7. Press the function menu key, then press the PRGRAM CHECK soft key.

8. You are now ready to single block your way through the program. Press the START button and have your hand over the feed hold at all times. Try to anticipate any interference between the tool point and the machine components. Press the feed hold button as soon as you think a problem will arise. You can always press the start button again and resume positioning.

Dry Run

The dry run will allow the spindle to rotate at the programmed speed rates yet still modify the rapid and feed rate positioning. Follow the same procedures above except place the TAPE CHECK/DRY RUN toggle switch to the DRY RUN position. Again, use caution and anticipation. Have your hand over the FEED HOLD button at all times. Observe that the spindle rotations occur in the proper direction and at the proper times. Check that the coolant is on when needed. If you have any questions during the operation of the machine, press the feed hold and call your instructor. Do not walk away from a machine that is in feed hold.

Error Recovery

Almost all programming errors will be revealed in the above prove out operations. Format errors, syntax errors, and axis over travel errors will

cause the machine to go into a feed hold condition. Call your instructor over if this happens. In some instances the operator will interrupt the program with a feed hold because they anticipate an interference is imminent. Whenever the above feed hold conditions occur we must determine what has caused the condition, safely reposition the tool to zero return, and correct the problem that was responsible for the error. Follow the procedures below whenever a problem causes the machine tool to enter a feed hold condition.

1. If a program error occurs an alarm message and error code will appear on the screen. A red flashing ALM will appear in the lower left corner of the display until the alarm is cancelled. Record the error code and the alarm message.

2. Press the CANCEL soft key to cancle the alarm.

3. Press the function menu key and then the PROGRAM soft key.

4. Record the block number or the entire block that contains the cursor. This block is usually the one that has the error in it. By looking at this block and the error message you should be able to determine how to fix the problem.

5. Sometimes the error is not so apparent and may be dependent on some program code that will be coming up since the control can look ahead several blocks in order to prepare for certain tool paths. Therefore, we may need more information to resolve the problem. Press the PRGRAM CHECK soft key.

6. Record the Absolute position for X and Z, and also the Dist To Go values for both X and Z. Also note or record the G codes which are in effect. This is necessary because this information will be lost once you reset the control, and it may be useful in determining what caused the error.

7. If the error message identified an axis over travel or limit error then you should also press the POSITION soft key and record the the (Machine) position in both X and Z.

8. Press the RESET button on the CRT panel.

9. Select the JOG mode.

10. Carefully position the tool point away from the part to a point where it can return home with out a problem.

11. Select the ZERO RETURN mode. Zero return both axes.

12. Select the EDIT mode. Edit the program or check your offsets and make the necessary corrections to eliminate the error condition.

13. Select the MEMORY mode. Press the PROGRAM soft key.

14. Press the REWIND soft key.

15. Rerun your prove out operation.

Trial Cut

The trial cut operation is where the part is finally loaded into the chuck and the feeds and speeds are verified for correctness. Certain combinations of materials, diameters, part overhang, tool extentions and feeds and speeds may result in harmonic vibrations which induce chatter in the machined surface of the part. Materials also vary in grades and alloys so feeds and speeds which may have run well with one type of aluminum, for example, may not work well at all on a different type of aluminum.

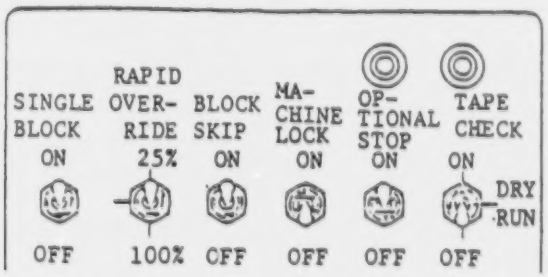
The trial cut is also where size verification is performed. Size variances

can occur due to: part and tool deflection, programming to the specified dimension when unilateral tolerancing was used, G50 values being rounded to the nearest one thousandth, or the depth of cut and tool pressure relation. In order to avoid scrapping the first workpiece it is a good idea to use the wear offsets to add a plus .010 to the X wear offset for each O.D. tool.

Some programs may have a built in trial cut as part of program for certain high tolerance diameters. These may be used where there is a lot of material removal and the wear on the insert can affect the size of the diameter from part to part. The OTIONAL STOP toggle switch should be on and optional stop codes (M01) should follow each turn diameter that requires a check.

The trial run is the first time that you will be running the program with a part. There is still a danger of tool collisions with the workpiece during rapid positioning and errant feed moves. For this reason the rapid traverse rate should still be limited to 5% or 25% to allow you time to react to an emergency situation. Use feed hold on most tool approaches and look at the program check screen to see if the distance to go values agree with what you observe. The single block should still be on so that you can read the next block and anticipate the correctness of the next tool movement. Follow the procedure below.

1. Set the toggle switches as shown.



2. Press the function menu key. Then press the OFFSET soft key.

3. Press the WEAR soft key.

4. Set the MEMORY LOCK/WRITE key to WRITE.

5. Cursor to the O.D. tool offset register.

6. Key in
   .01

7. Press the INPUT soft key. Do not press the +INPUT soft key.

8. Do steps 5 through 7 for each O.D. tool.

9. Select MEMORY mode.

10. Press the function menu key. Press the PROGRAM soft key.

11. Press the operation menu key. Press the REWIND soft key.

12. Press the function menu key. Press the PRGRAM CHECK soft key.

13. Press the program start key each time for single block operation. Use the feedrate override and spindle speed override to control incorrect feeds and speeds.

    If the horsepower is exceeding 100% then reduce the sindle speed first. If it is still exceeding then reduce the feed rate also. Once you have reduced the feed rate you can try increasing the spindl speed again and monitor the effect on horsepower.

    If the tool is chattering then first increase the feed rate. If it still chatters then try reducing the spindle speed.

Record these override settings for each tool. Set the overrides back to 100% before the next tool is used.

14. After each M01 is read, an optional stop will occur. Mic the diameter that was just machined and record the difference as shown below.

    (wear offset) = (print spec) - (mic reading)

    This value will be either positive (if it was machined too small) or negative (if it was machined too large). Since we already have .010 in the wear offset register our wear offset should calculate to -.010 if everything in the program and setup was exactly perfect.

15. While the machine is still in the optional stop call up the wear offset screen. Cursor to the wear offset register for the current tool. Key in the calculated wear offset from step 14. Don't forget to key in the negative sign if it was negative.

16. This time press the +INPUT soft key.

17. Call up the program check screen. Continue the program and repeat steps 14 through 16 for each tool.

18. After you have completely run the program then select the EDIT mode. For each tool you had to use the overrides on calculate the new feed rate or spindle speed by multiplying the override rate by the program values to get the new value and edit the program accordingly.

You are now ready to run the program at full automatic.

ASSIGNMENT SECTION

Section 1

1-1. What kind of information does the CNC documentation provide?

1-2. What does the G50 command do?

1-3. Describe where the values for the X and Z words in a G50 command block are derived from.

1-4. Why should you use a paper feeler when you touch off tools on the part face?

1-5. When setting the G50 Xvalue for boring bars you add 17.0 inches to twice the F dimension of the boring bar. Where does the 17.0 inches come from?

Section 2

2-1. Name four different methods of running a CNC part program. These are typically used for program prove out.

2-2. When tape check is on are the X and Z axes frozen during program execution?

2-3. During dry run at what speed does the rapid traverse travel at?

2-4. What happens when an alarm signal starts flashing on the screen?

2-5. How would you clear the alarm?

2-6. During the trial run you find that some of the diameters are a few thousandths oversize. How do you correct this problem?

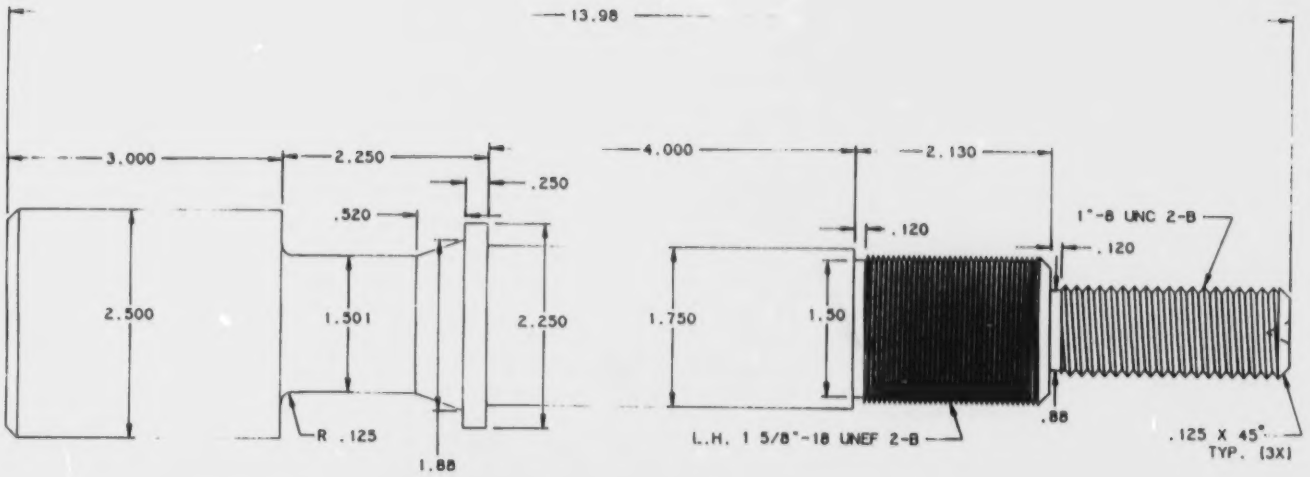
PROJECT SECTION

Observe the demonstration project.

Competency Evaluation

DATE	DESCRIPTION
_____	Set the G50 X and Z values for a turning tool
_____	Prepare the machine for a tape check
_____	Clear an alarm condition

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations



TOLERANCES	COURSE : 444-332	matc	PART NO.
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL : ALUM.		NC 332.0
	DATE DRAWN : 1/3/90		
	DRAWN BY:B. RAMCKE		SCALE= 1:1

24

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

TURNING CENTER TOOL LIST

Part Name	Part No.	Unit No.
DEMO	NC 332.0	4
	Operation No.	Program No. O32Ø1

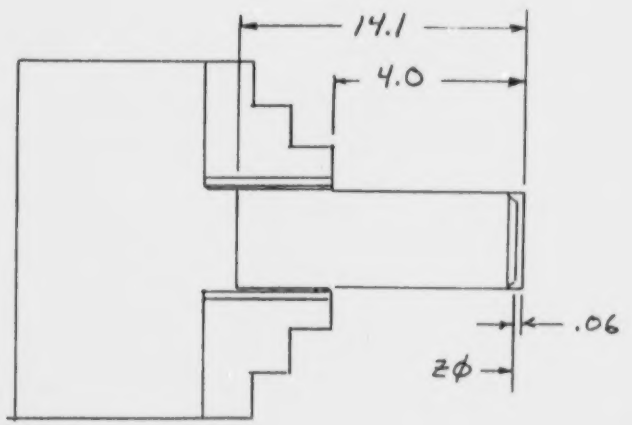
Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. Ft/Min.	R.P.M.	Feed I.P.R.
1	T8	DDJNR-164C Rgh Turn KV45-ETAL 16H		08		600		.012
2	T4	DVJNR-164D Fin Turn B-119875-R01		04		600		.012
3	T1	NSR-163C (NG3189R) Grv KV45-ETAL-16AH		01		600		.010
4	T2	NSR-163C (NTP-3R) RH Thrd KV45-ETAL-16AH		02			800	.125
5	T2	SAME ABOVE LH Thrd		02			800	.0556
6	T4	SAME ABOVE UnderCut RH		05		1500		.012
7	T6	DVJNL-164D UnderCut LH KV45-ETAL-16H		06		1500		.012

T1	T2	T3	T4	T5
T6	**T7**	**T8**	**T9**	**T10**

matc
444 - TURNING CENTER SETUP SHEET

Part No. NC 332.ϕ	Part Name DEMO PART	Mch. No.	Unit No. 4
Program No O 332ϕ	Holding Device 3-Jaw Chuck w/ Hard Jaws		
Manual Operation #1	Face, Chamfer, and Turn Left End (3.0 X 2.5 Dia)		

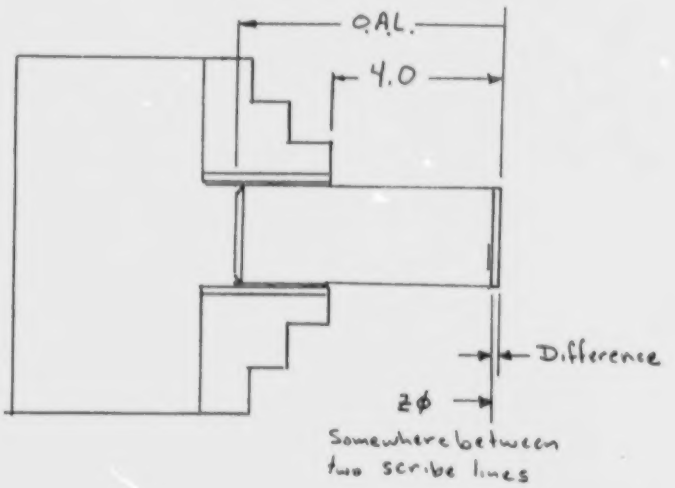
Raw Stock 2.50 D. x 14.1
Skin Cut 2.5 dia for 3.25 length



matc

444 - TURNING CENTER SETUP SHEET

Part No. NC 332.Ø	Part Name DEMO PART	Mch. No.	Unit No. 4
Program No. O332Ø	Holding Device 3-Jaw Chuck w/ Hard Jaws		
Manual Operation #2	Face and Center Drill Right End		



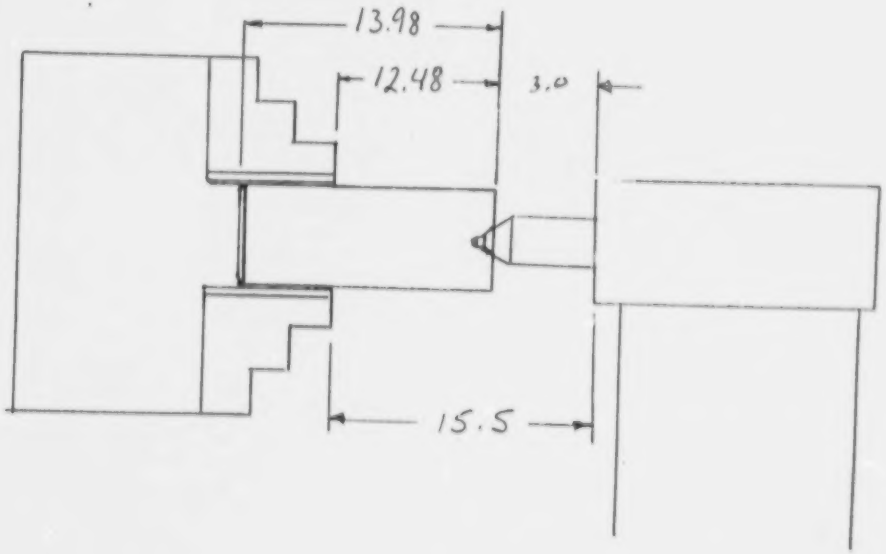
Scribe a line on the O.D. at 13.975 and at 13.985 from the finished face. Note the over all length of the part at this point. When you face off the face use the difference between the O.A.L. and 13.985 to add to your G50 value.
Face to 13.98 Dim. and Center Drill to 5/16 D.a.

matc

444 - TURNING CENTER SETUP SHEET

Part No. NC 332.Ø	Part Name DEMO PART	Mch. No.	Unit No. 4
Program No. O332Ø	Holding Device 3-Jaw Chuck w/ Soft Jaws		

Machine soft Jaws per SJ3321-6 ; Machine Part Complete

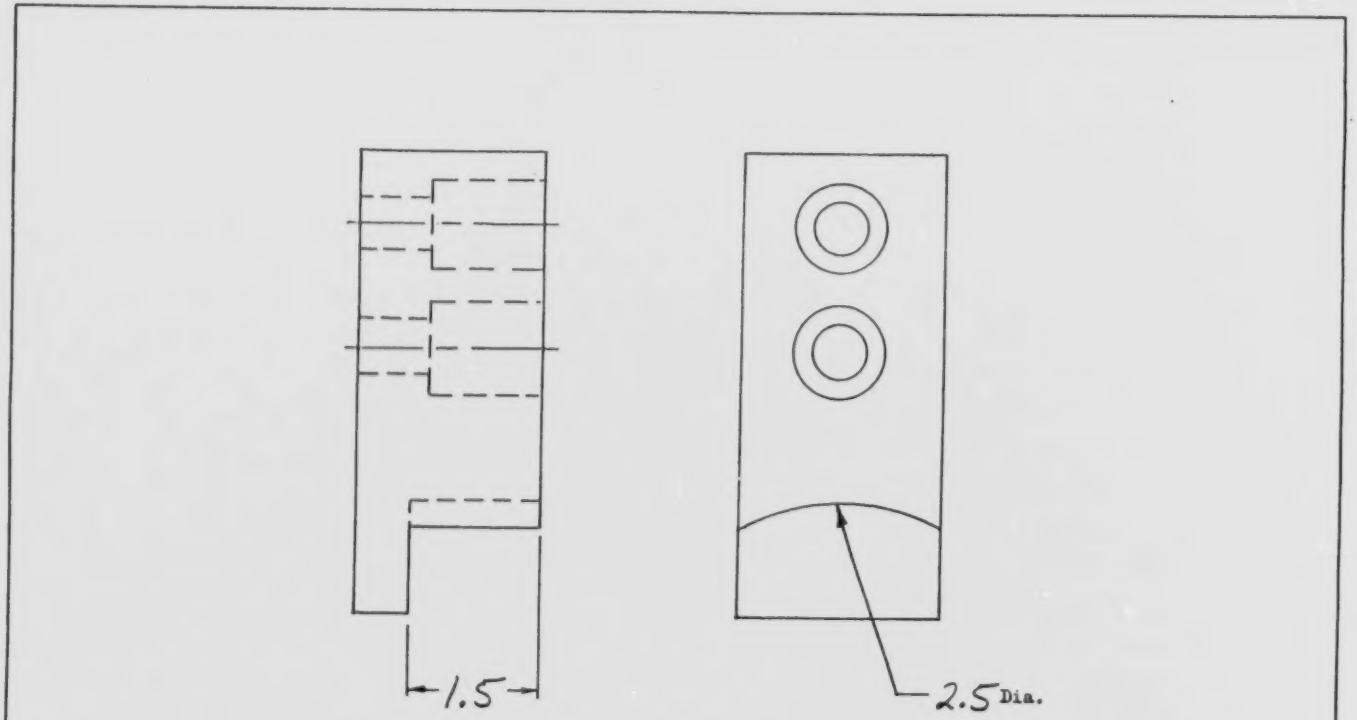


OFFSET WORKSHEET

Part No.	Part Name	Mch. No.	Unit No.

Work Zero Offset (Shift)		Tool Length Offset — Geometry/G50					Radius Compensation Offset		
		Tool	Offset				Insert	Tool Tip	
X	Z	No.	WEAR	GEO.	X	Z	Radius	No.	Notes

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations



Course			MILWAUKEE AREA TECHNICAL COLLEGE	
Tolerances X = ± .010 XX = ± .005 XXX = ± .002	Material Purchased soft jaw blanks	Drawn	Part Name SOFT JAWS	
			Micro-Inch Finish 125	Part No. SJ 3321-6

CNC PROGRAM

Part No. NC332.0	Part Name DEMO	Program No. O3201	Unit No. 4

Part Program	Program Description
% O3201(DEMO TN GRV & THRD) ; N10G20 ; N20G50X9.928Z10.435S2500 ; N30T0808M08; N40M41 ; N50G96S600M03; N60G00Z.1; N70X2.6; (CANNED RGH TURN CYCLE); N80G71P90Q220U.01W.01I.01K.01 D0500F.012, N90G00X.75 ; N100G01Z0F.01; N110X1.Z-.125; N120Z-2.59 ; N130X1.375 ; N140X1.625Z-2.725;	

Program No.	Program Description
N150Z-4.725; N160X1.75; N170Z-8.73 ; N180X2.25; N190Z-10.98; N200X2.48; N210X2.5Z-10.99; N220X2.6 ; N230G00X9.928Z10.435T0800M09 ; N240M01; N250G50X9.903Z10.419S2500; N260G00T0404M08; N270M41; N280G96S600M03 ; N290G00Z.1 ; N300X2.6; (CANNED FINISH TURN CYCLE) ; N310G70P90Q220 ; N320X3.; N330G00X9.903Z10.419T0400M09 ;	

Program No.	Program Description
N340M01;	
(GRV);	
N350G20;	
N360G50X9.7Z10.425S2500;	
N370T0101M08 ;	
N380M41;	
N390G96S600M03 ;	
N400G00Z.1 ;	
N410X2.;	
N420Z-2.6;	
N430G01X.88F.01;	
N440G04U.5 ;	
N450G01Z-2.595F.01 ;	
N460G00X2.25 ;	
N470Z-4.73 ;	
N480G01X1.5F.01;	
N490G04U.5 ;	
N500G01Z-4.725F.01 ;	
N510G00X3.5;	
N520G00X9.7Z10.425T0100M09 ;	

Program No.	Program Description
N530M01; (1-8 UNC 3A THRD); N540G20; N550G50X9.488Z10.511S800 ; N560T0202M08 ; N570M40; N580G97S800M03 ; N590G00Z.1 ; N600X1.01; (CANNED THREAD CYCLE); N610G76X.8492Z-2.5K.0754A60 D100F.125 ; N620G00X3. ; N630G00X9.488Z10.511T0200M09 ; N640M01; (1-5/8 UNEF LH THRD) ; N650G20; N660G50X9.488Z10.511S800 ; N670T0202M08 ; N680M40; N690G97S800M03 ;	

Program No.	Program Description
N700G00Z.1 ; N710X2.0 ; N720Z-4.58; (CANNED THREAD CYCLE) ; N730G76X1.5287Z-2.5I0K.048 A60D100F.0556; N740G00X3.0; N750G00X9.488Z10.511T0200M09 ; N760M01; (UNDERCUT RH); N770G20; N780G50X9.903Z10.419S2500; N790T0405M08 ; N800M41; N810G96S1500M03; N820G00Z-8.98; N830X2.5 ; N840G01X2.25F.012; N850X2.15Z-9.073 ; N860Z-10.975 ;	

Program No.	Program Description
N870X2.3 ;	
N880G00Z-9.073 ;	
N890G01X2.15F.012;	
N900X2.05Z-9.156 ;	
N910Z-10.975 ;	
N920X2.25;	
N930G00Z-9.156 ;	
N940G01X2.05 ;	
N950X1.95Z-9.239 ;	
N960Z-10.975 ;	
N970X2.1 ;	
N980G00Z-9.239 ;	
N990G01X1.95 ;	
N1000X1.85Z-9.322;	
N1010Z-10.975;	
N1020X1.95 ;	
N1030G00Z-9.322;	
N1040G01X1.75Z-9.405 ;	
N1050Z-10.975;	
N1060X1.85 ;	

Program No.	Program Description
N1070G00Z-9.405;	
N1080G01X1.75F.012 ;	
N1090X1.65Z-9.488;	
N1100Z-10.97 ;	
N1110G02X1.75Z-10.975R.125 ;	
N1120G01X1.8F.012;	
N1130G00Z-9.488;	
N1140G01X1.65F.012 ;	
N1150X1.55Z-9.571;	
N1160Z-10.930;	
N1170G02X1.65Z-10.980R.125 ;	
N1180G01X1.7F.012;	
N1190G00Z-9.571;	
N1200G01X1.55F.012 ;	
N1210X1.5Z-9.613 ;	
N1220Z-10.855;	
N1230G02X1.75Z-10.980R.125 ;	
N1240G01X2.5F.01 ;	
N1250G00X9.903Z10.419T0400M09;	
N1260M01 ;	

Program No.	Program Description
(UNDERCUT LH);	
N1270G20 ;	
N1280G50X9.712Z11.913S2500 ;	
N1290T0606M08;	
N1300M41 ;	
N1310G96S1500M03 ;	
N1320G00Z.1;	
N1330Z-9.98;	
N1340X2.0;	
N1350G01X1.7F.02 ;	
N1360X1.54F.012;	
N1370X1.52Z-9.605;	
N1380Z-9.491 ;	
N1390X2.122Z-8.99;	
N1400X2.3;	
N1410G00Z-9.98 ;	
N1420X1.7;	
N1430G01X1.54F.012 ;	
N1440X1.52Z-9.491;	
N1450X1.722Z-9.326 ;	

Program No.	Program Description
N1460X2.004Z-8.99; N1470X2.2; N1480G00Z-9.8; N1490X1.75 ; N1500Z-9.38; N1510G01X1.722Z-9.326F.012 ; N1520X1.894Z-8.99; N1530X2.1; N1540G00Z-9.9; N1550X1.7; N1560G01X1.51F.01; N1570X1.5Z-9.5 ; N1580X1.88Z-8.98 ; N1590X2.26 ; N1600G00X3.0 ; N1610G00X9.712Z11.913T0600M09; N1620M30 ; %	

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 5

Tooling

INTRODUCTION

This unit will introduce you to a new and unique concept in lathe tooling systems. Traditionally only 25% of a CNC lathe's available time is used for machining. The other 75% is non-machining time roughly divided into 60% set-up time and 15% non-cutting motions, such as machine loading and positioning of the tool out of cut. The unique Quick Setter and Quick Change Tool features reduce set-up time to an estimated 35%

OBJECTIVES

At the completion of this unit the student will be able to:

1. Assemble tools into the proper holders
2. Describe the Quick Change Tooling System advantages
3. Setup and machine projects demonstrating face and turn operations.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

CONTENTS

Reference Section — Page No.

Section 1 Quick Change Tooling System — 5 -

Assignment Section

On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section. — 5 -

Project Section

Project NC332.1 — 5 -
Project NC332.2

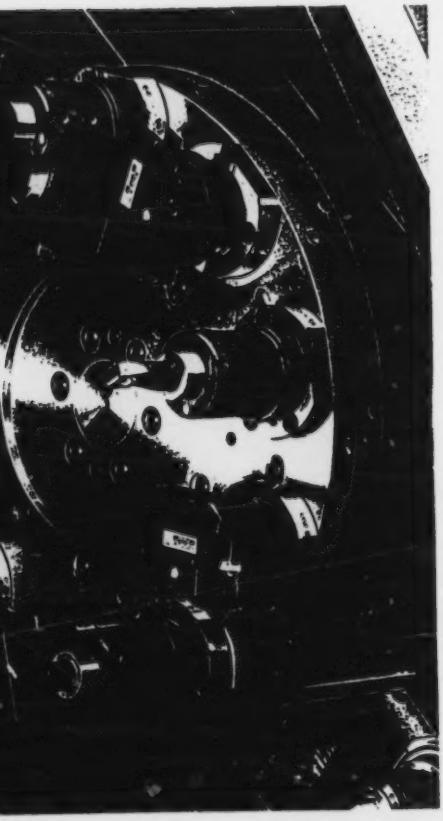
SECTION 1

QUICK CHANGE TOOLING SYSTEM

The unique Quick Change Tooling System si
time. A tool is changed in five seconds. Wre
rigid tool holder has a clamping force of one
Just flip a switch and the tool is easily remo
tooling is so universal that any turret station
tools. Tools for the next job can be prepare
machine to continue operation.

ations

System significantly reduces set-up
nds. Wrenches are not necessary. The
ce of one ton, yet operation is easy.
asily removed from the turret. The
et station accepts either I.D. or O.D.
e prepared off the machine allowing the

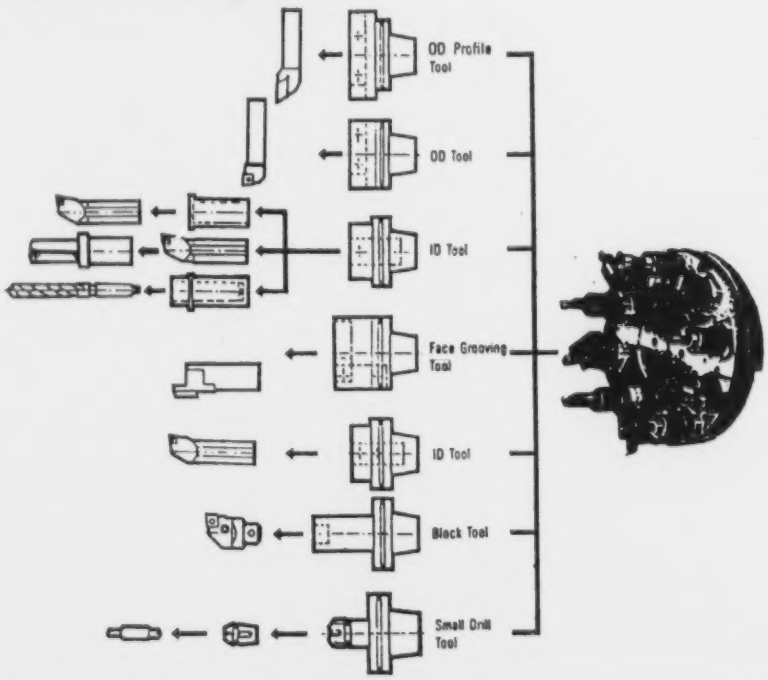


BEST COPY AVAILABLE

The turret features an easy loading procedure that was covered in a previous unit.

The turret head employs the KV tooling systems. Study the diagram below which illustrates the combinations of tooling for the various tooling assemblies. Actual tool assemblies are shown on the next page.

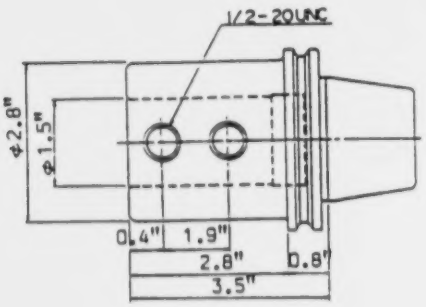
Quick Change Tooling System



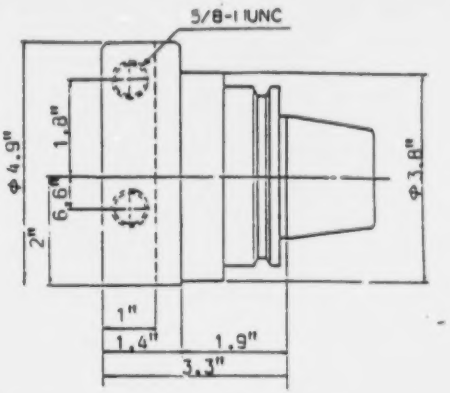
BEST COPY AVAILABLE

BEST COPY AVAILABLE

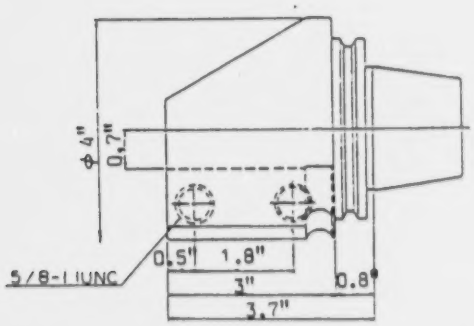
There are several styles of both O.D. and I.D. tool holders. The I.D. tools provide several different lengths for boring heads, boring bars and collet adapters for drilling, reaming, and tapping. There are three types of O.D. tool holders as shown below.



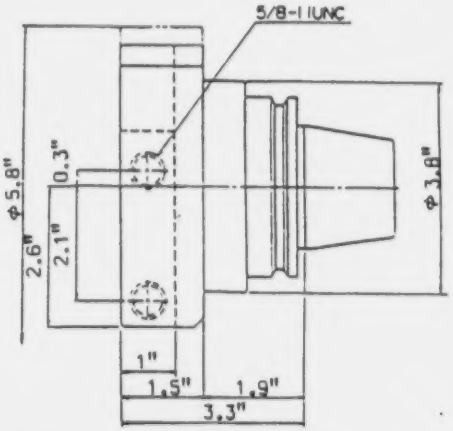
I.D. Boring Bar Holder



O.D. Face And Turn Tool Holder

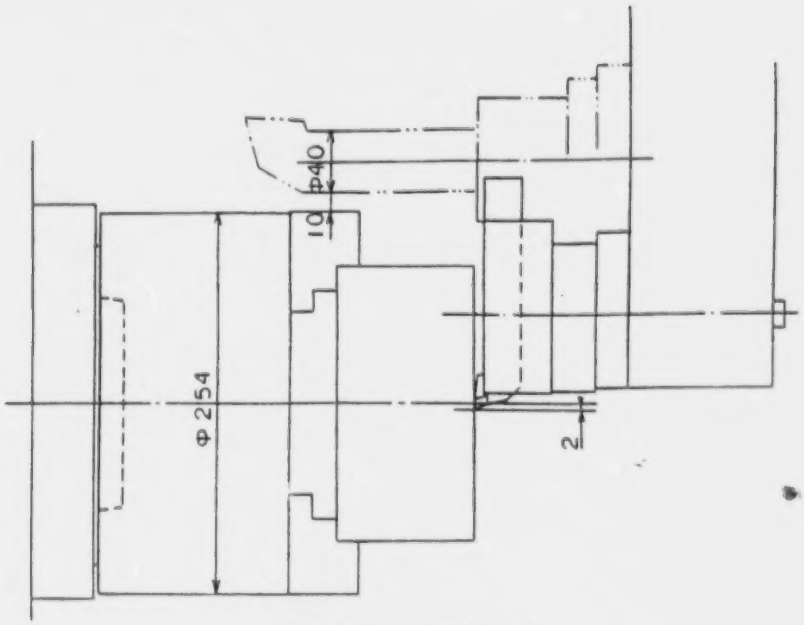


O.D. Trepan Tool Holder

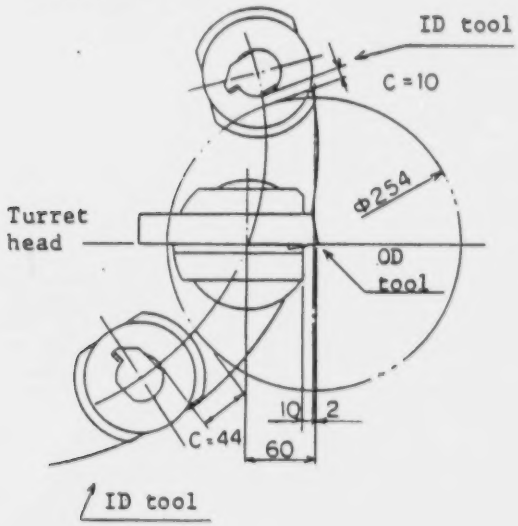


O.D. Copying Tool Holder

When both O.D. and I.D. tools are used in a setup or unused turret stations still have tools in them it is recommended that O.D. and I.D. tools alternate in the turret positions so that I.D. tools do not interfere with the chuck or jaws. Even when they are alternated a facing tool can bring an I.D. tool into the chuck or jaws. Make sure the jaws are set as close as possible to the chuck circumference. The following page illustrates this relation.



Relations Between
Chuck/Jaws and Tool



Tool Layout
(OD-ID)

PROJECT SECTION

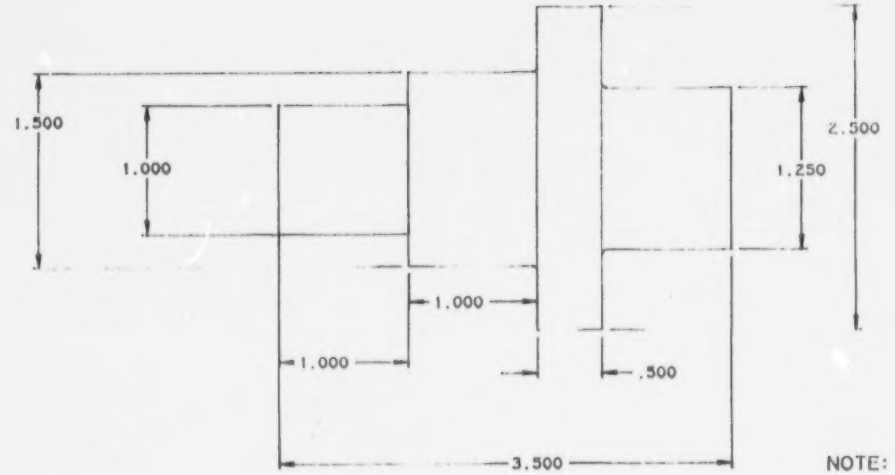
Setup and machine the following two projects: NC332.1 and NC332.2. Each project will require the use of a special machined soft jaw and both projects have two operations. The actual machining is very simple, the exercise here is to become familiar with the machine handling in setup, proveout and operation.

Competency Evaluation

DATE	DESCRIPTION
_____	Setup and machine NC332.1
_____	Setup and machine NC332.2

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

NOTE: BREAK ALL EDGES



NOTE: .032 RADIUS MAX.

TOLERANCES	COURSE 444332		PART NO.
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL ALUM	matc	NC 332.1
	DATE DRAWN 1/3/90		
	DRAWN BY R RACH		SCALE= 1:1

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

TURNING CENTER TOOL LIST

Part Name	Part No. NC 332.1	Unit No.
	Operation No. 1 & 2	Program No. 03211 & 03212

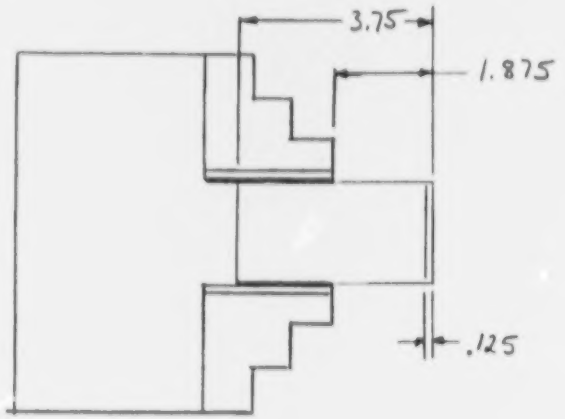
Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. FtMin.	R.P.M.	Feed I.P.R.
1	T8	DCLNR-164C Rgh Turn KV45-ETAL-16H		08		600		.012
2	T8	SAME ABOVE Fin Turn		08		600		.012

T1	T2	T3	T4	T5
T6	**T7**	**T8**	**T9**	**T10**

matc

444 - TURNING CENTER SETUP SHEET

Part No. NC 332.1	Part Name	Mch. No.	Unit No.
Program No. 03211	Holding Device Hard Jaws Open #1		
	Material: 3.0 Dia X 3.75		

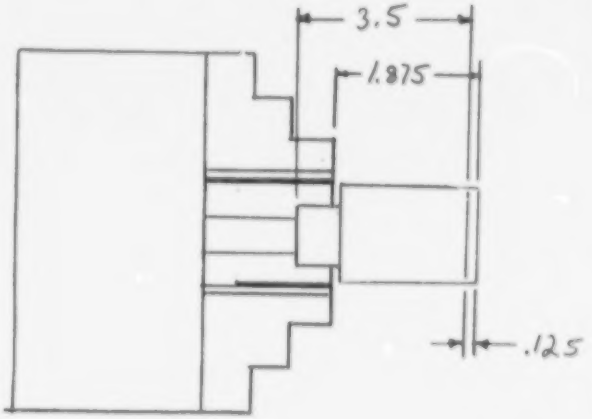


MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

11

matc
444 - TURNING CENTER SETUP SHEET

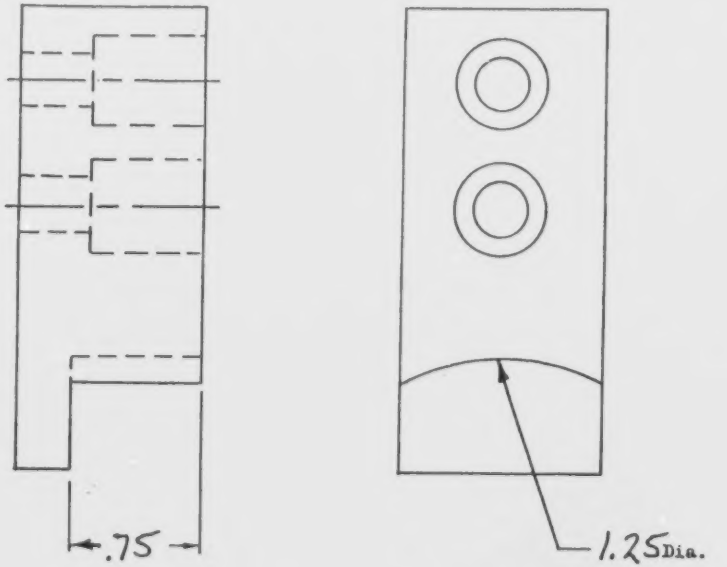
Part No. NC 332.1	Part Name	Mch. No.	Unit No.
Program No. 03212	Holding Device SJ3322 (Machine Soft Jaws per Print) Oper #2		



OFFSET WORKSHEET

Part No.	Part Name	Mch. No.	Unit No.

Work Zero Offset (Shift)		Tool Length Offset — Geometry/G50					Radius Compensation Offset		
		Tool	Offset				Insert	Tool Tip	
X	Z	No.	WEAR	GEO.	X	Z	Radius	No.	Notes



Course			MILWAUKEE AREA TECHNICAL COLLEGE
Tolerances X = ± .010 XX = ± .005 XXX = ± .002	Material Purchased soft jaw blanks	Drawn	Part Name SOFT JAWS
			Micro-Inch Finish 125 / Part No. SJ 3322

CNC PROGRAM

Part No. NC 332.1	Part Name	Program No. 03211	Unit No.

Part Program	Program Description
%	
O3211 (OP. #1 - NC332.1);	
N10G20 (TOUCH OFF AT JAW - MOVE Z+1.875);	
N20G50X10.232Z20.984S3000;	
N30T0808M08 (DCLNR - 164C);	
N40M41 (INSERT - CNMP 432 KC 910);	
N50G96S600M03;	
N60G00Z.1;	
N70X3.2;	
N80G01X0.F.012;	
N90G00Z.2X3.2;	
N100Z0.;	
N110G01X0.F.012;	
N120G00Z.1;	
N130X3.2;	
(CANNED TURN CYCLE);	
N140G71P150Q220U.01W.01I.01K.01 D1000F.015;	

Program No.	Program Description
N150G00X1.2188; N160G01Z0.F.01; N170X1.25Z-.0156; N180Z-1.0; N190X2.4688; N200X2.5Z-1.0156; N210Z-1.625; N220X3.2; N230G00X10.232Z20.984T0800M09; N240M01; (CHECK DIA'S 1.25 & 2.5); N250G5010.232Z20.984S3000; N260G00T0808M08; N270M41; N280G96S600M03; N290G00Z.1; N300X3.2;	

Program No.	Program Description
(CANNED FINISH TURN CYCLE); N310G70P150Q220F.01; N320G00X3.5; N330G00X10.232X20.984T0800M09; N340M30; %	

CNC PROGRAM

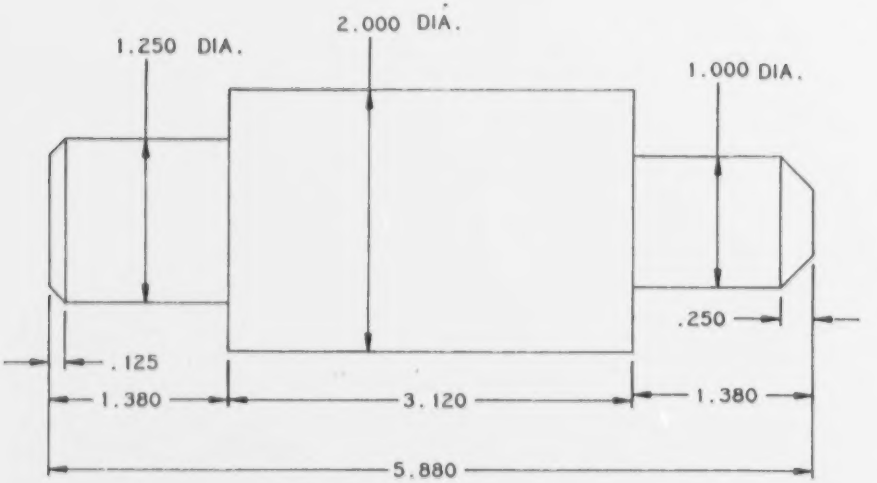
Part No. NC 332.1	Part Name	Program No. O3212	Unit No.

Part Program	Program Description
%; O3212; (OP. #2 - NC3321); N10G20; (TOUCH OFF AT JAW FC - MOVE Z+2.625); N20G50X10.Z20.S3000; N30T0808M08 (DCLNR-164C); N40M41 (INSERT- CNMP432 KC910); N50G96S600M03; N60G00Z.1; N70X3.2; N80G01X0.F.012; N90G00X3.2Z.2; N100Z0.; N110G01X0.F.012; N120G00Z.1; N130X3.2;	

Program No.	Program Description
(CANNED RGH TURN CYCLE); N140G71P150Q250U.01W.011.01K.01 D0500F.012; N150G00X.9688; N160G01Z0F.01; N170X1.Z-.0156; N180Z-1.; N190X1.467; N200X1.5Z-1.016; N210Z-2.; N220X2.467; N230X2.5Z-2.016; N240Z-2.125; N250G00X3.2; N260G00X10.Z20.T0800M09; N270M01; (CHECK DIA'S 1.00 & 1.5); N280G50X10.Z20.S3000; N290T0808; N300M41;	

Program No.	Program Description
N310G96S600M03; N320G00Z.1; N330X3.2; (CANNED FINISH TURN CYCLE); N340G70P150Q250F.01; N350G00X3.5; N360G00X10.Z20.T0800M09; N370M30; %	

NOTE: CHAMFER ON BOTH ENDS 45 DEGREES
NO. 5 DRILL AND CSINK BOTH ENDS



TOLERANCES	COURSE : 444-332		PART NO.
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL : ALUM.	matc	NC 332.2
	DATE DRAWN : 1/4/90		SCALE= 1:1 CV RUN>NC332.2
	DRAWN BY: B.RAMCKE		

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

TURNING CENTER TOOL LIST

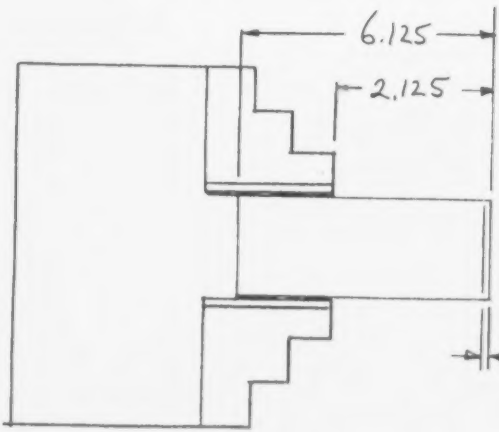
Part Name	Part No.	Unit No.
PIN	NC332.2	
	Operation No. 1 & 2	Program No. 03221 & 03222

Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. Ft/Min.	R.P.M.	Feed I.P.R.
1	T10	DCLNR-164C Right Turn KV45-ETAL-164		10		600		.012
2	T7	#5 Center Drill KV45 SU 1½		07		600		.010
3	T10	SAME ABOVE Fin Turn		10		600		.012
		2nd Operation						
1	T10	SAME ABOVE Right Turn		10		600		.010
2	T8	DDJNR-164C Fin Turn KV45-ETAL 164		08		600		.010

T1	T2	T3	T4	T5
T6	**T7**	**T8**	**T9**	**T10**

444 - TURNING CE

Part No. NC332.2	Part Name
Program No. 03221	Holding Device Hard Jaws
	Mat'l: 2.5 O.D. x 6.



23

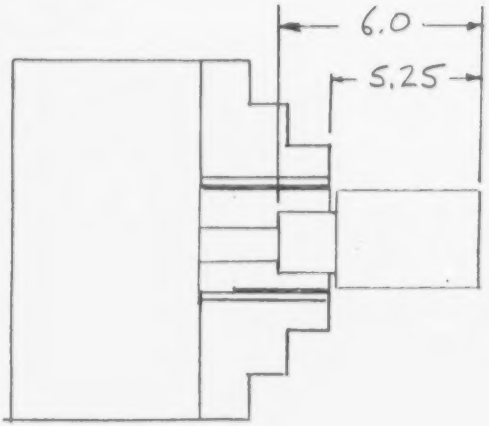
MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

atic
NTER SETUP SHEET

Mch. No.	Unit No.
Op. #1	
25	

444 - TURNING CE

Part No. NC 322.2	Part Name
Program No. 03222	Holding Device SJ 332L



24

atc
NTER SETUP SHEET

Mch. No.	Unit No.
Oper #2	

OFFSET W

Part No.	Part Name	M

Work Zero Offset (Shift)		Tool Length Offset — Geometr				
		Tool No.	Offset			
X	Z		WEAR	GEO.	X	

25

RKSHEET

n. No.	Unit No.

/G50	Radius Compensation Offset		
Z	Insert Radius	Tool Tip No.	Notes

CNC PROGRAM

Part No. NC332.2	Part Name	Program No. O3221	Unit No.

Part Program	Program Description
%; O3221 (PREP OPER. FC & CTR); N10 G20 (SET PART OUT 2.125 FROM JAW FC.); N20G50X10.Z20.S3000; (TOUCH OFF JAW FC & MOVE 2.00 Z+); N30T1010M08 (DCLNR-164C); N40M41 (INSERT CNMP 432 KC 910); N50G96S600M03; N60G00Z.05; N70X2.6; N80G01X0.F.012; N90G00X2.8Z.1; N100G01Z0.F.012; N110X0.; N120G00X3.Z.2; N130G00X10.Z20.T1000M09; N140M01;	

Program No.	Program Description
N160G50X0Z20S3000;	
N170T0707M08 (#5 CENTER DRILL);	
N180M41;	
N190G96S600M03;	
N200G00Z.1;	
N210G97S445M03;	
N220X0.;	
N230G01Z-.4F.01;	
N240G04X.5;	
N250G00Z.5;	
N260X3.;	
N270G00X0.Z20.T0700M09;	
N280M00;	
(TURN PART AROUND -SET OFF JAW 2.125);	
N290G20;	
N300G50X10.Z20.S3000;	
N310T1010M08;	
N320M41;	
N330G96S600M03;	
N340G00Z.05;	

Program No.	Program Description
N350X2.6; N360G01X0.F.012; N370G00X2.8Z.1; N380G01Z0F.012; N390X0.; N400G00Z.2; N410X2.6; (CANNED TURN CYCLE RGH); N420G71P430Q490U.01W.01I.01 K.01D0500F.012; N430G00X1.; N440G01Z0.F.012; N450X1.250Z-.125; N460Z-1.380; N470X1.9375; N480X2.Z-1.4425; N490X2.6; N500G00X10.Z20.T1000M09; N510M01; N520G50X10.Z20.S3000;	

Program No.	Program Description
N530G00T1010M08; N540M41; N550G96S600M03; N560G00Z.2; N570X2.6; N580G70P430Q490F.01; N590G00X3.; N600G00X10.Z20.T1000M09; N610M30; %	

CNC PROGRAM

Part No. NC332.2	Part Name	Program No. O3222	Unit No.

Part Program	Program Description
%; O3222 (PIN SECOND OPER); (SET PC OUT 5.25 FROM JAW FC); N10G20X10.Z20.S3000; N30T1010M08 (DCLNR-164C); N40M41 (INSERT-CNMP 432 KC 910); N50G96S600M03; N60G00Z.1; N70X2.6; N80G71P70Q160U.01W.011.01 K.01D0500F.012; N90G00X.75; N100G01Z0F.01; N110X1.0Z-.125; N120Z-1.38; N130X2.; N140Z-4.6; N150X2.6;	

Program No.
N160G00X10.Z20.T1000M09;
N170M01;
(CHECK DIA'S 1.00 & 2.00);
N180G50X10.Z20.S3000;
N190T0808M08;
N200M41;
N210G96S600M03;
N220G00Z.1;
N230X2.6;
N240G70P90Q160F.010;
N250G00X2.7;
N260G00X10.Z20.T0800M09;
N270M30;
%

s

Program Description

PROJECT SECTION

Setup and machine the following two projects: NC332.3 and NC332.4. The first project involves a steep taper and a partial arc. The second project will challenge you with a different measuring unit. The program and part are in metric.

Competency Evaluation

DATE	DESCRIPTION
_____	Setup and machine NC332.3
_____	Setup and machine NC332.4

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 6

Canned Cycle Machining

INTRODUCTION

With canned cycle machining complex par
once in the CNC part program and then in
cycle G codes that are available. In this
and the finish turn operation are only one
to the part geometry are also safely mod
blocks. Due to international business opp
be called upon to machine a part which is
G code will command the turning center
moves in units of millimeters. Since 1m
take some getting use to when trying to i
actual position values when proving out

OBJECTIVES

At the completion of this unit the student

1. Setup and proveout part program
geometries programmed with canne
2. Describe canned cycle codes G70
3. Setup and proveout part program
geometries programmed with canne
measurement.
4. Recognize the G-codes for inch

rt geometries can be described
voked or run with several canned
way the roughing turn operation
block in length. Any corrections
dified in the geometry description
portunities for job shops you may
s dimensioned in metric. A simple
to perform positional and feed
mm is approximately .040" it will
magine the distance to go and
the metric part programs.

t will be able to:
ns of parts with complex
d cycles.
and G71.
ns of parts with complex
d cycles and in units of metric

and metric positioning units.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

CONTENTS

Reference Section	Page No.
Section 1 Canned Cycle Machining	6 -
Section 2 Metric Machining	6 -
Assignment Section	
On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section.	6 -
Project Section	
Project NC332.3	6 -
Project NC332.4	

SECTION 1

CANNED CYCLE MACHINING

The following is an excerpt from the part program for project NC332.3, the ~ indicates where the program has been removed for space saving. Refer to the entire program is at the end of this unit for details.

```
N300G00Z.1;                                    <-- (Upper Left of
Raw Stock)
N310X4.2;                                      <--        "
N320G71P330Q400U.01W.01I.01K.01D0500F.015;     <-- (Rough Turn
Operation)
N330G00X1.75;                                  <-- (Part Geometry
Description)
     ~                                         <--        "
N400X4.2;                                      <--        "
     ~
N490G70P330Q400F.01;                           <-- (Finish Turn
Operation)
     ~
```

The G71 is a canned cycle for rough turning and rough boring. It will be covered in more detail in the programming manual. Before the G71 is invoked for a turn operation the programmer must position the tool point to the upper right corner of the stock. This point tells the controller how much stock must be removed to get down to the part geometry defined later. This is used to compute the number of roughing passes required along with some other information provided in the G71 block. The P word provides the line number (N word) of the block that begins to define the part geometry, and the Q word is the block containing the end of the part geometry description. This geometry description should start on the next block after the G71 block.

The U word is the finish allowance left on the diameter or X coordinates. The W word is the finish allowance for the Z values. The I and K words are the semi-finish allowances for the X and Z values. The D word is the depth of cut for the roughing passes. The F word is the feed rate. The feed rate can be respecified in the part geometry blocks. If it is not then roughing passes can be programmed at a higher rate than the finish pass that will be specified in the finish turn canned cycle.

The G70 block is the canned cycle for the finish turn or finish bore operations. The G70 block should specify the same P and Q words that were specified in the G71 block and a new feed rate if required.

SECTION 2

METRIC MACHINING

A metric part will be dimensioned in millimeters. When compared to the same part dimensioned in inch units the metric part will have numbers of higher magnitude and it may take some getting use to. For example all programming words which are written in inch units will have to be converted into millimeter units. This includes such commands as feed rates and finish allowances in canned cycles, involving X, Z, U, W, I, K, D, and F words. There are 25.4 mm per 1in, and this can be used as a conversion factor.

INCH			MILLIMETER
X8.75	x	25.4	= X222.25
F.006	x	25.4	= F.152
U.015	x	25.4	= U.381

The G20 code which is used at the beginning of all the part programs run so far is the G code used to tell the controller the unit of measure is in inches. The metric program will use G21 to indicate that the program is in metric or units of measure in millimeters.

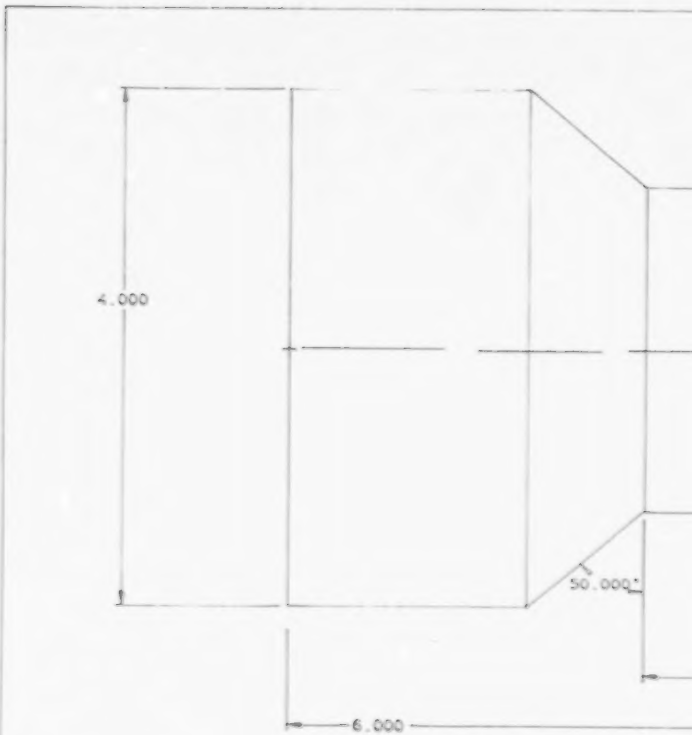
PROJECT SECTION

Setup and machine the following two projects: NC332.3 and NC332.4. The first project involves a steep taper and a partial arc. The second project will challenge you with a different measuring unit. The program and part are in metric.

Competency Evaluation

DATE	DESCRIPTION
_____	Setup and machine NC332.3
_____	Setup and machine NC332.4

ISi



TOLERANCES	COURSE 444332
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL ALUM
	DATE DRAWN 1/4/90
	DRAWN BY R RACH

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

NICAM III: SAMPLE 1

matc	PART NO. NC 332.3
	SCALE= 1: 1

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

TURNING CENTER TOOL LIST

Part Name	Part No. NC 332.3	Unit No.
	Operation No. 1	Program No. O3231

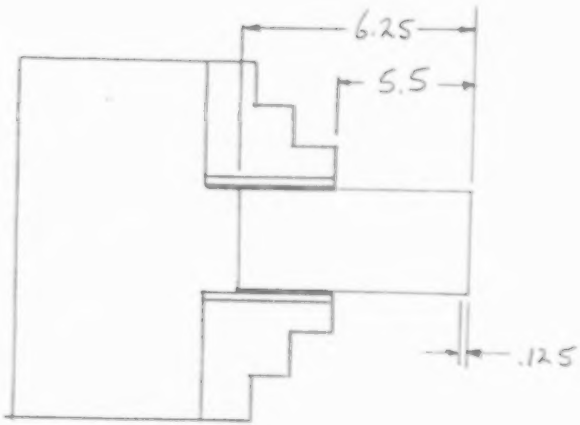
Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. Ft/Min.	R.P.M.	Feed I.P.R.
1	T10	DCLNR-164C Fc & Rgh Turn KV-ETAL-16H		10		600		.012
2	T10	SAME ABOVE Fc & Rgh Turn		10		600		.012
3	T8	DDJNR-164C KV45-ETAL-16H		08		600		.012

T1	T2	T3	T4	T5
T6	**T7**	**T8**	**T9**	**T10**

m

444 - TURNING CE

Part No. NC 332.3	Part Name
Program No. 03231	Holding Device SJ 3323
	Matl: 4" O.D x 6.25



6

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

atc
NTER SETUP SHEET

	Mch. No.	Unit No.

OFFSET WOR

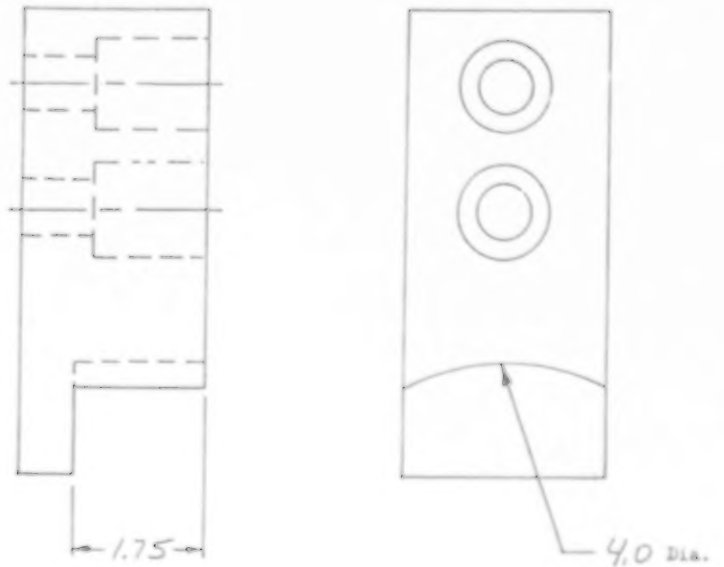
Part No.	Part Name	Mch.

Work Zero Offset (Shift)		Tool Length Offset — Geometry				
		Tool No.	Offset		X	
X	Z		WEAR	GEO.		

RKSHEET

No.	Unit No.		
G50	Radius Compensation Offset		
Z	Insert Radius	Tool Tip No.	Notes

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations



Course			MILWAUKEE AREA TECHNICAL COLLEGE	
Tolerances X = ± .010 XX = ± .005 XXX = ± .002	Material Purchased soft jaw blanks	Drawn	Part Name SOFT JAWS	
			Micro-inch Finish 125	Part No. SJ3323

150

CNC PROGRAM

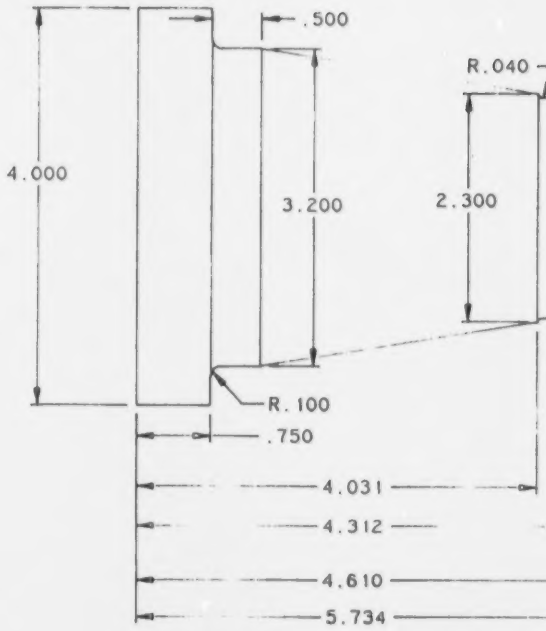
Part No. NC 332.3	Part Name	Program No. 03231	Unit No.

Part Program	Program Description
%; O3231 (TWO PART OPER.); (SOFT JAWS 4" ID); (SET PART OUT 5.5 FROM JAW FC); N010G20 (SET TOOL 5.375 OFF JAW FC); N020G50X10.Z20.S3000; N030T1010M08; N040M41; N050G96S600M03; (FC .125 OFF & TURN TO 3.9375 DIA); N060G00Z.05; N070X4.2; N080G01X0.F.012; N090G00X4.2Z.2; N100Z.01; N110G01X0.F.012; N120Z0.;	

Program No.	Program Description
N130X3.9068; N140X3.9375Z-.0625; N150Z-2.; N160G00X4.2; N170G00X10.Z20.T1000M09; N180M00 (PROGRAM STOP); (REVERSE PART); (SET PC OUT 4.375 FROM JAW FC); N190G20; N200G50X10.Z20.S3000; N210T1010M08; N220M41; N230G96S600M03; (FC TO 6.00 LENGTH); N240G00Z.05; N250X4.2; N260G01X0.F.012; N270G00X4.2Z.2; N280G01Z0.F.010; N290X0.;	

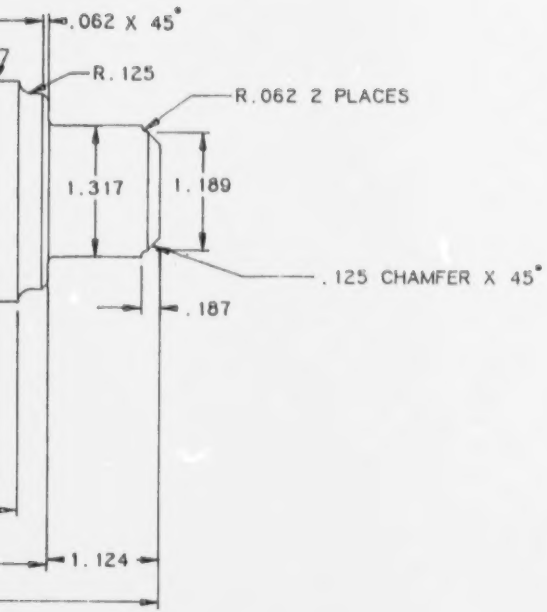
Program No.	Program Description
N300G00Z.1; N310X4.2; N320G71P330Q400U.01W.011.01 K.01D0500F.015; N330G00X1.75; N340G01Z0.F.012; N350X2.Z-.125; N360Z-1.941; N370G02X2.5Z-2.5R0.75; N380G01Z-3.25; N390X4.0Z-4.144; N400X4.2; N410G00X10.Z20.T1000M09; N420M01; N430G50X10.Z20.S3000; N440G00T0808M08; N450M41; N460G96S600M03; N470G00Z.1; N480X4.2;	

Program No.	Program Description
N490G70P330Q400F.01; N500G00X4.3; N510G00X10.Z20.T0800M09; N520M30; %	



TOLERANCES	COURSE: 444-332	
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL: ALUM.	
	DATE DRAWN: 1/4/90	
	DRAWN BY: B.RAMCKE	

91



	PART NO. NC 332.4
SCALE= 1: 1	CV RUN>NC332.4

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

TURNING CENTER TOOL LIST

Part Name	Part No. NC 332.4	Unit No.
	Operation No. 1	Program No. O 3241

Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. Ft/Min.	R.P.M.	Feed I.P.R.
1	T10	DCLNR-164C Rgh Turn KV45-ETAL-16H		10		580		.012
2	T8	DDJNR-164C Fin Turn KV45-ETAL-164		08		600		.012

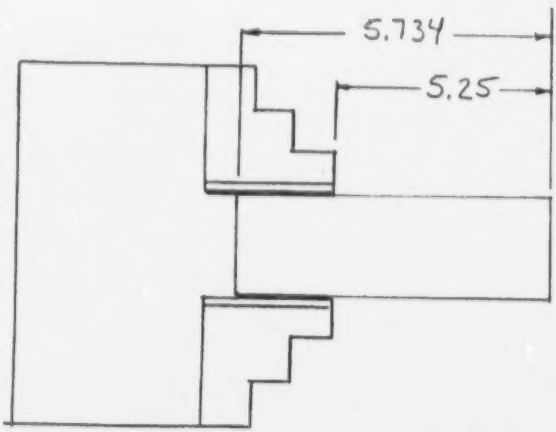
T1	T2	T3	T4	T5

T6	T7	T8	T9	T10

m

444 - TURNING CE

Part No. NC 332.4	Part Name
Program No 03241	Holding Device Hard Jaws
	Matl: 4.0



18

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

atc
NTER SETUP SHEET

Mch. No.	Unit No.

Note: Face & Chamfer both ends
to Length: 5.734
Then setup per sketch.

OFFSET WO

Part No.	Part Name	Mc

Work Zero Offset (Shift)		Tool Length Offset — Geometry				
		Tool No.	Offset			
X	Z		WEAR	GEO.	X	

200

RKSHEET

n. No.	Unit No.

/G50	Radius Compensation Offset		
Z	Insert Radius	Tool Tip No.	Notes

CNC PROGRAM

Part No. NC 332.4	Part Name	Program No. O3241	Unit No.

Part Program	Program Description
%; O3241; N10G20; N20G50X9.9Z10.4S3600; N30T1010M08; N40M41; N50G97S525M03; N60G96S580; N70G00Z.2; N80X4.2; N90G71P100Q270U.01W.01I.01 K.01D0500; N100G00X.939; N110G01Z0Z.012; N120X1.189Z-.125; N130Z-.179; N140G02X1.317Z-.241R0.062; N150G01Z-1.068;	

Program No.	Program Description
N160G02X1.441Z-1.124R0.062;	
N170G01X1.751;	
N180X1.875Z-1.186;	
N190Z-1.297;	
N200G02X2.125Z-1.422R0.125;	
N210G01X2.24;	
N220Z-1.703;	
N230X2.3;	
N240X3.2Z-4.484;	
N250Z-4.884;	
N260G02X3.4Z-4.984R0.1;	
N270G01X4.1;	
N280G00Z.2;	
N290G00X9.9Z10.4T1000M09;	
N300M01;	
N310;	
N320G50X9.93Z10.435S1075;	
N330G00T0808M08;	
N340M41;	

Program No.	Program Description
N350G97;	
N360G96S600;	
N370G00X6.Z2.;	
N380Z.3;	
N390X4.;	
N400G70P100Q270 ;	
N410G00X6.Z2.;	
N420G00X9.93Z10.345T0800M09;	
N430M30;	
%;	

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

Unit 7

Introduction

INTRODUCTION

This unit will introduce you to the setup and operation of part programs which require the use of the tailstock to provide support and rigidity for operations such as threading and grooving. It also provides a project which has a semi-circular nose end requiring the fine size adjustment of wear offsets.

OBJECTIVES

At the completion of this unit the student will be able to:

1. Setup and proveout part programs of parts with grooving and threading operations involving the use of the tailstock.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

CONTENTS

Reference Section — Page No.

Section 1 Tailstock

Assignment Section

On a separate sheet of paper provide answers for any questions found at the end of each section. For best results in learning and comprehension answer the questions before continuing on to the next section. 7 -

Project Section

Project NC332.5 7 -
Project NC332.6

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

SECTION 1

TAILSTOCK

The second project for this unit requires the use of the tailstock. During the setup the part will require facing operations to obtain the finished part length and a center drill operation for the use of the center. The center and tailstock should be setup according to the instructions given in the setup sheet. The describes how to operate the tailstock and its precautions.

Precautions

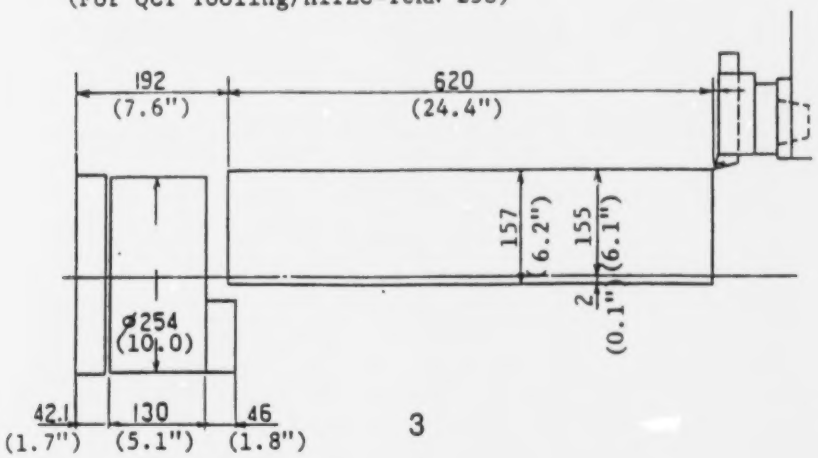
1. When the tailstock is not in use move the tailstock to the retract end of its stroke.

2. Do not use a stationary center in the tailstock.

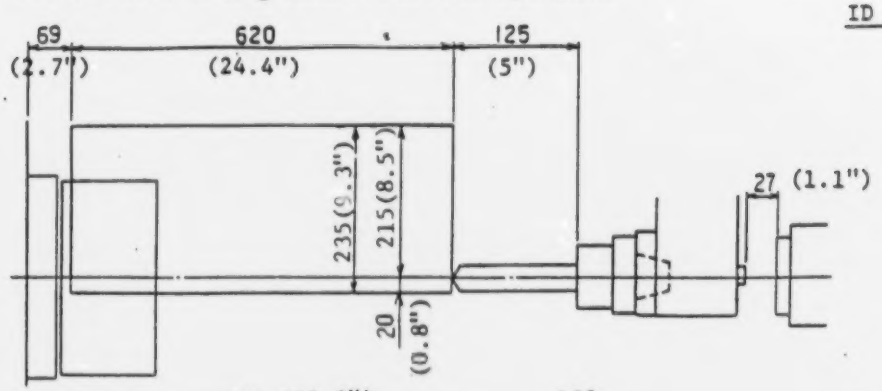
3. In center work, pay careful attention to an interference between the tool and the rotary center, and also between the turret head body and the quill or tailstock proper. Diagrams below illustrate the full range of O.D. and I.D. tools and the tailstock and their relation in interference conditions.

Tool Traveling Range
(For QCT Tooling/HITEC-TURN 25S)

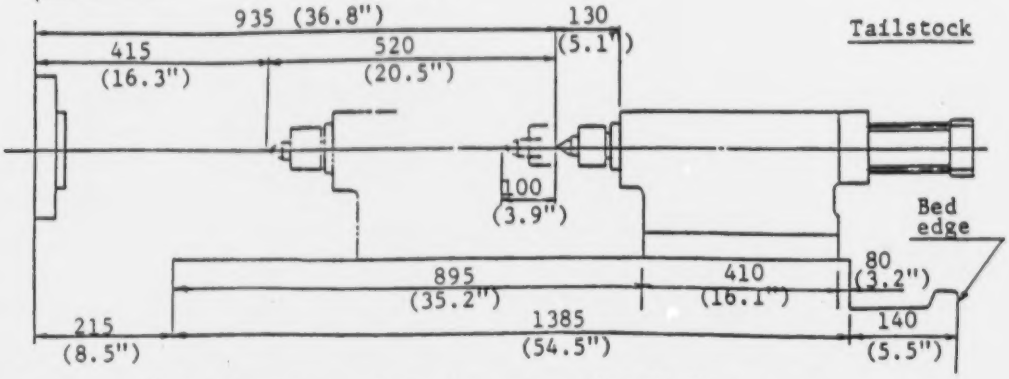
OD Tool



ID Tool



Tailstock



Machining range when the center (LC-5F-6) is used.

(1) OD roughing tool
(standard projection)

(2) Copying tool (Note) The tool is projected 40 mm more than a standard one.

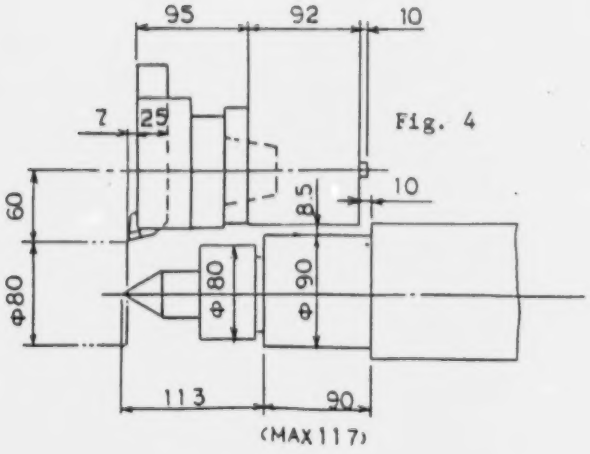
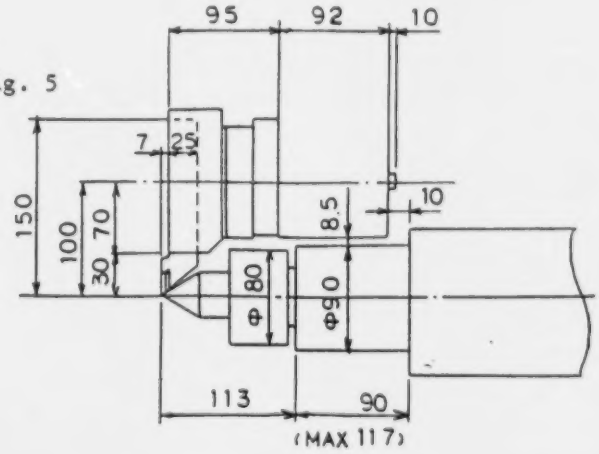


Fig. 4

Fig. 5



Setup and Operation

1. Set the hydraulic pressure of the tailstock spindle. Relation between the pressure and thrust of the tailstock are as shown below. Apressure control valve is located on the upper part of the hydraulic unit in the rear of the machine.

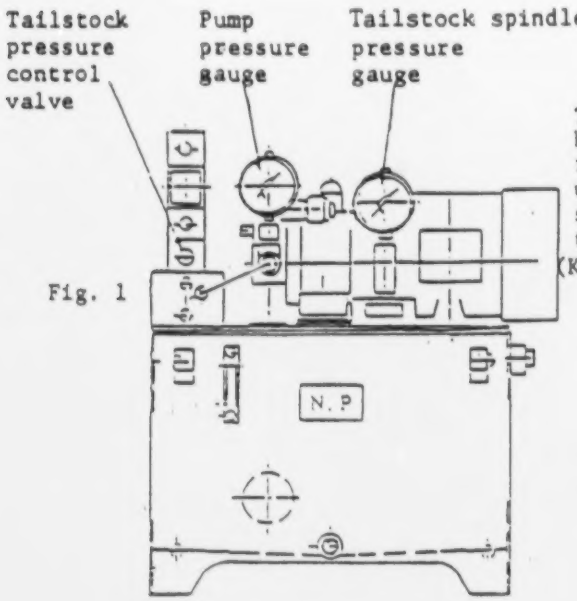


Fig. 1

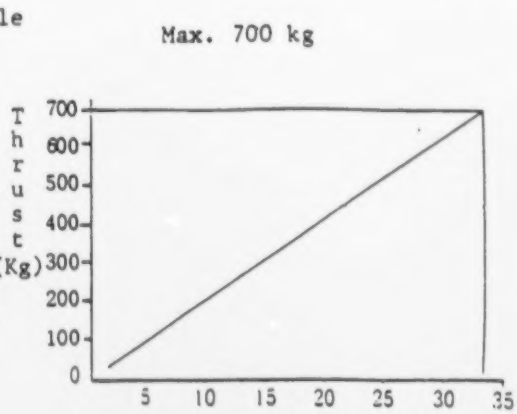
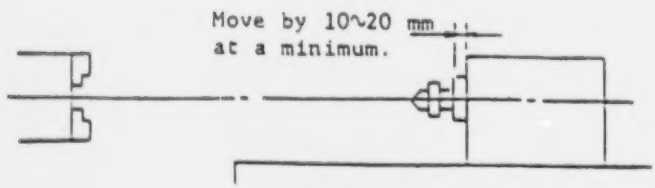


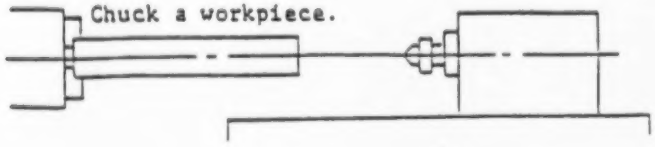
Fig. 2

A thrust reference value is about 200 kg when a blank of ø100 × 300 (equivalent to S45C) is used.

2. Jog the tailstock spindle forward and stop it before the retract end.

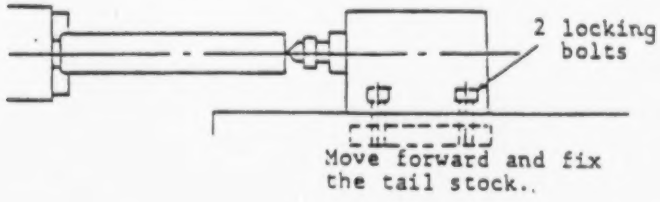


3. Chuck the work piece.

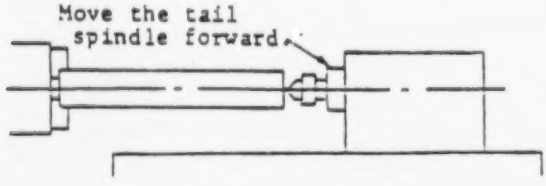


4. Move the tailstock by hand until the center comes into the center hole of the workpiece.

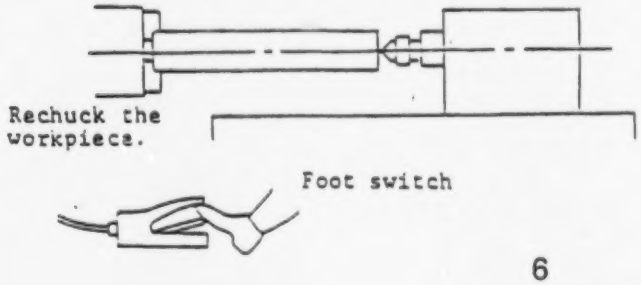
5. Fix the tailstock at that position. Tighten two clamping bolts.



6. Move the tailstock spindle forward by pressing the CENTER FORWARD button. The green light above this button will come on.



7. Press the chuck open/close foot switch several times, little by little, to rechuck the workpiece.

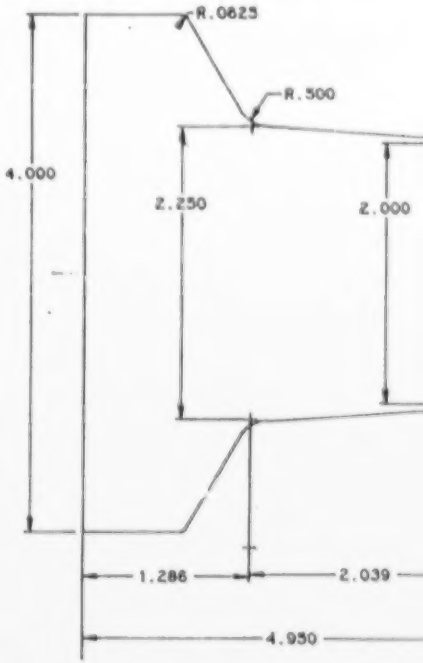


PROJECT SECTION

Setup and machine the following two projects: NC332.5 and NC332.6.

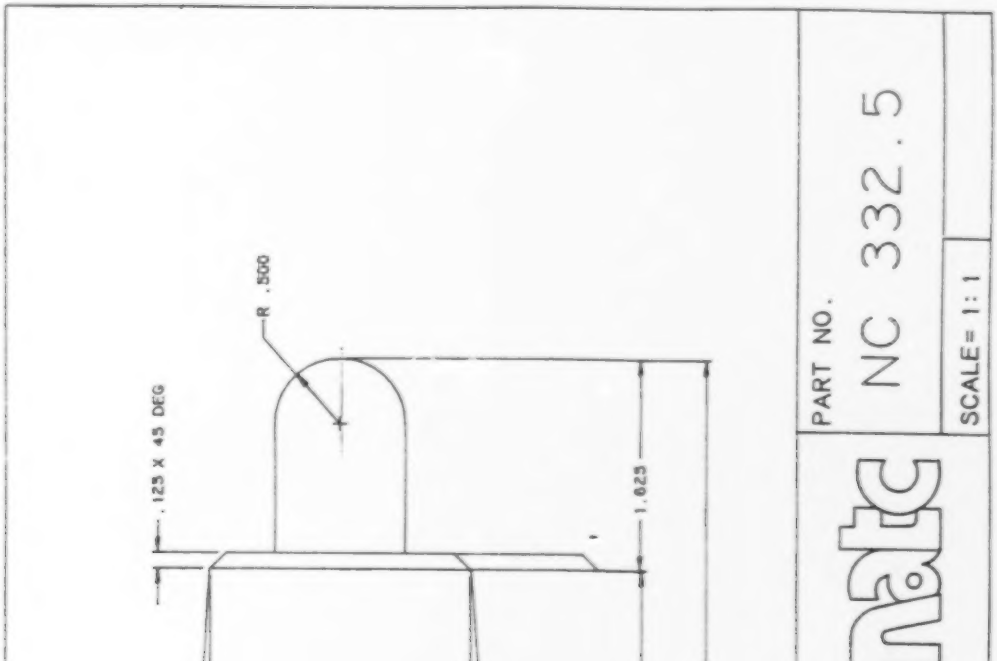
Competency Evaluation

DATE	DESCRIPTION
_____	Setup and machine NC332.5
_____	Setup and machine NC332.6



TOLERANCES	COURSE: 444-332
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL: ALUM.
	DATE DRAWN: 1/4/90
	DRAWN BY: R RACH

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations



MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

TURNING CENTER TOOL LIST

Part Name	Part No. NC332.5	Unit No.
	Operation No. 1	Program No. 03251

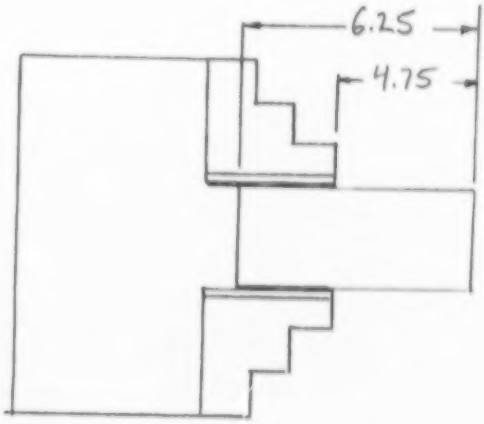
Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. Ft/Min.	R.P.M.	Feed I.P.R.
1	T10	DCLNR-164C RghTurn KV45-ETAL-16H		10		600		.012
2	T10	Same Above RghTurn		10		600		.012
3	T8	DDJNR-164C FinTurn KV45-ETAL16H		08		600		.012

T1	T2	T3	T4	T5
T6	**T7**	**T8**	**T9**	**T10**

m&
444 - TURNING CEN

Part No. NC 332.5	rt Name
Program No. 02251	Holding Device SJ 3323

Matl: 4" OD. x 6.2



10

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

atc
ITER SETUP SHEET

Mch. No.	Unit No.
5	

OFFSET WORKSHEET

Part No.	Part Name	Mch. No.	Unit No.

Work Zero Offset (Shift)		Tool Length Offset — Geometry/G50					Radius Compensation Offset		
		Tool No.	Offset				Insert Radius	Tool Tip No.	Notes
X	Z		WEAR	GEO.	X	Z			

CNC PROGRAM

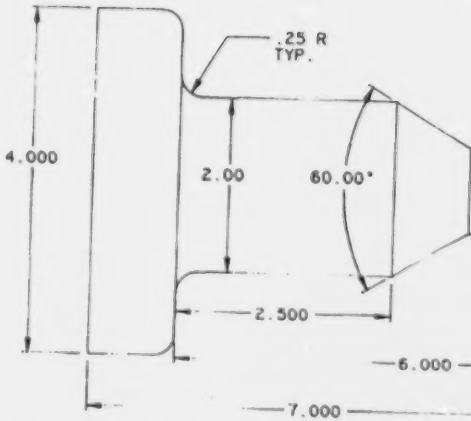
Part No. NC 332.5	Part Name	Program No. 03251	Unit No.

Part Program	Program Description
%; O3251 (ONE OPER); (FC, TN & CONTOUR); N010G20; (SOFT JAWS 4" ID); (SET PART 4.75 FROM JAW FC); (SET TOOL 4.5625 FROM JAW FC); N020G50X10.Z20.S3000; N030T1010M08; N040M41; N050G96S600M03; N060G00Z.06; N070X4.2; N080G01X0.F.012; N090G00X4.2Z.2; N100Z.01; N110G01X0.F.012;	

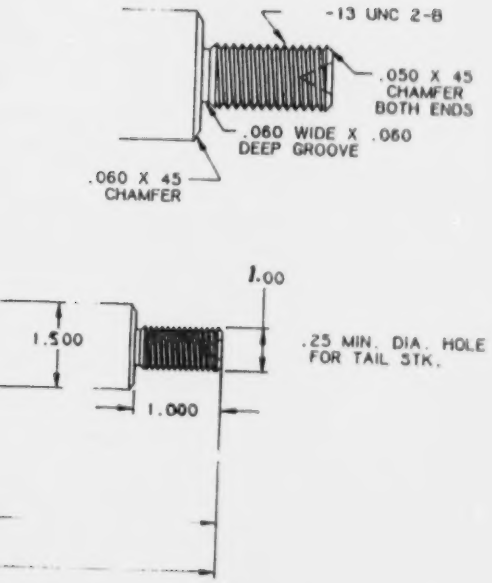
Program No.	Program Description
N120Z0.F.01;	
N130X3.9068;	
N140X3.9375Z-.0625;	
N150Z-2.5;	
N160G00X4.2;	
N170G00X10.Z20.T1000M09;	
N180M00;	
(REVERSE PART);	
(SET PART 4.75 FROM JAW FC);	
N190G50X10.Z20.S3000;	
N200T1010M08;	
N210M41;	
N220G96S600M03;	
N230Z.06;	
N240X4.2;	
N250G01X0.F.012;	
N260G00X4.2Z.2;	
N270Z0.;	
N280G01X-.03F.01;	
N290G00Z.2;	

Program No.	Program Description
N300X4.2; N310G71P320Q430U.01W.01I.01 K.01D0500F.012; N320G00X-.03; N330G01Z0.F.012; N340G03X1.25Z-.625R.625; N350G01Z-1.5; N360X1.75; N370X2.Z-1.625; N380X2.218Z-3.396; N390G02X2.716Z-3.798R.5; N400G01X3.9374Z-4.151; N410G03X4.Z-4.205R.0625; N420G01Z-4.3; N430X4.2; N440G00X4.5; N450G00X10.Z20.T1000M09; N460M01; N470G50X10.Z20.S3500; N480G00T0808M08;	

Program No.	Program Description
N490M41; N500G96S600M03; N510G00Z.2; N520X4.2; N540G70P320Q430; N560G00X4.5; N570G00X10.Z20.T0800M09; N580M30; %	



TOLERANCES	COURSE : 444-332	
∠=±1/2° .0=±.015 .00=±.005 .000=±.001 .0000=±.0005	MATERIAL : ALUM.	
	DATE DRAWN : 1/4/90	
	DRAWN BY : B.RAMCKE	



matc	PART NO. NC 332.6
	SCALE= 1: 1 / CV RUN>NC332.6

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

matc

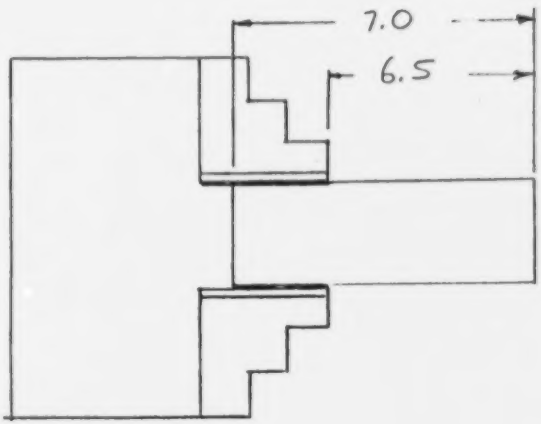
TURNING CENTER TOOL LIST

Part Name	Part No. NC332.6	Unit No.
	Operation No. 1	Program No. 03261

Tool Seq. No.	Tool No.	Tool Description	Tool Dia.	Tool Offset No.	Tool Radius	Surf. Ft/Min.	R.P.M.	Feed I.P.R.
1	T8	DDJNR-164C Rgh Turn KV45-ETAL-164		08		600		.012
2	T4	DVJNR-164D Fin Turn B-118875-R01		04		600		.012
3	T1	NSR-163C (NG3189R) Groove KV45-ETAL-16AH		01		600		.006
4	T2	NSR-163C (NTP-3R) Thread KV45-ETAL-16AH		02			800	.0769

T1	T2	T3	T4	T5
T6	**T7**	**T8**	**T9**	**T10**

Part No. NC 332.6	Part Name	M No
Program No 03261	Holding Device Hard Jaws	
	Matl: 4" O.D. x 7.7	



18

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

tC
TER SETUP SHEET

ch.	Unit No.
5"	

Note: Face & chamfer both ends to Length 7.0. Centerdrill one end then setup per drawing.

MILWAUKEE AREA TECHNICAL COLLEGE
Industrial Division
444-332 CNC Turning Center Advanced Operations

OFFSET WORKSHEET

Part No.	Part Name	Mch. No.	Unit No.

Work Zero Offset (Shift)		Tool Length Offset — Geometry/G50					Radius Compensation Offset		
		Tool No.	Offset				Insert Radius	Tool Tip No.	Notes
X	Z		WEAR	GEO.	X	Z			

CNC PROGRAM

Part No. NC332.6	Part Name	Program No. 0326/	Unit No.

Part Program	Program Description
%; O3261(AXLE TN GRV & THRD); N10G20; N20(CANNED CYCLE ROUGH TURN); N30G50X9.93Z10.435S2500; N40T0808; N50M41; N60G96S600M03; N70G00Z.1; N80X4.2; N90G71P100Q220U.01W.01I.01 K.01D1000F.012; N100G00X.75; N110G01Z0F.012M08; N120X1.0Z-.125; N130Z-1.0; N140X1.38; N150X1.5Z-1.06;	

Program No.	Program Description
N160Z-3.067; N170X2.0Z-3.5; N180Z-5.75; N190G02X2.5Z-6.0R.25; N200G01X3.5; N210G03X4.0Z-6.25; N220G01X4.2; N230G00X9.93Z10.435T0800M09; N240M01; ; N250(CANNED CYCLE FINISH TURN); N260G50X9.9Z10.42S2500; N270G00T0404; N280M41; N290G96S600M03; N300G00Z.1; N310X4.2; N320G70P100Q220; N330X4.2G00X9.9Z10.42T0400M09;	

Program No.	Program Description
N340M01; ; N350(GROOVE); N360G50X9.7Z10.425S2500; N370T0101; N380M41; N390G96S600M03; N400G00Z.1; N410X1.7Z-1.0; N420G01X.88F.006M08; N430G04U.5; N440G00X1.2; N450Z-.84; N460G01X.88Z-1.0F.004; N470G00X1.7; N480X9.7Z10.425T0100M09; N490M01; ; N500(1-13 UNC 3B THRD);	

Program No.	Program Description
N510G50X9.49Z10.51S800; N520T0202; N530M40; N540G97S800M03; N550G00Z.1; N560X1.01; N570G76X.9Z-.95K.0499A60 D100F.0769; N580G00X3.0; N590X9.49Z10.51T0200M09; N600M30; %	